
Users Guide To

RMA2 WES Version 4.3

**US Army Corps of Engineers - Waterways Experiment Station
Hydraulics Laboratory**

This manual and on-line documentation were produced using *Doc-To-Help*®, by WexTech Systems, Inc.



WexTech Systems, Inc.
310 Madison Avenue, Suite 905
New York, NY 10017
(212) 949-9595



FastTABS is a product of the Engineering Computer Graphics Laboratory, Brigham Young University.

Users Guide To RMA2 WES Version 4.3

Last document modification date: May 19, 1997

Effective RMA2 source code modification date: October 1, 1996

Editor:	<i>Barbara P. Donnell</i>
Web Layout Editor:	<i>W. Clay LaHatte</i>
Original Author:	<i>Dr. Ian King</i>
Contributing Authors:	<i>Barbara P. Donnell, Dr. John I. Finnie, Joseph V. Letter, Jr., William. H. McAnally, Jr., Dr. Lisa C. Roig, and William A. Thomas</i>
Source Code Authors:	<i>Barbara P. Donnell, Dr. John I. Finnie, Dr. Ian King, Joseph V. Letter, Jr., and Dr. Lisa C. Roig</i>
Technical Assistance:	<i>Dr. R. C. Berger, Jr., Robert A. Evans, Jr., Dr. Gary E. Freeman, and Dr. Robert T. McAdory, Jr.</i>

Technical Support

World Wide Web: <http://hlnet.wes.army.mil>

E-mail: tabs@hl.wes.army.mil

Contents

Overview	1
What Is RMA2.....	1
Origin Of The Program	1
Applications For RMA2	1
Capabilities Of RMA2.....	1
Limitations Of RMA2	2
Governing Equations.....	3
Element Types Supported.....	4
Two Dimensional Elements.....	5
One Dimensional Elements	5
Special Elements	6
Curved Element Edges	8
What's New And What's Gone	8
Ch- Ch- Ch- Ch- Changes	9
Additions	9
Deletions	10
System Requirements	10
Personal Computer Systems.....	10
Mini Computer Systems (Workstations)	11
Mainframe/Supercomputer Systems.....	11
 Using RMA2	 13
The RMA2 Modeling Process	13
Modes Of Operation.....	14
Running In Interactive Mode.....	14
Running In Batch Mode	16
Guidelines For Obtaining A Good Solution	16
Maintain Good Element Properties.....	16
Maintain Good Mesh Properties	17
Match Resolution With The Situation.....	17
Pre-RMA2 Simulation Check List	17
Persistent Divergence	18
 Basic Operation	 19
Data Files.....	19
Run Control Files	19
Geometry File.....	20
Hotstart File	20
Results Listing Files	20
Solution File.....	21
Vorticity Solution File	22

Using Titles	22
Including Title Information	22
The Last Title Card	22
What Kind Of Computer Do You Have?	23
Hey, What Do You Care?	23
So What Do You Want Me To Do About It?	23
Specifying Initial Conditions	23
Coldstart Initial Conditions	24
Hotstart Initial Conditions	24
Iteration Control	24
What Is An Iteration?	24
How RMA2 Finds A Solution	24
Regulating The Number Of Iterations	25
Now, What Do I Really Do?	27
Convergence Problems	27
Time Step Control	28
Selecting A Time Step Interval	28
Setting the Computational Time Interval	28
Total Simulation Time	29
Steady State Simulation Time	29
Dynamic Simulation Time	29
Simulating With Tides	29
Specifying Boundary Conditions	29
Providing Dynamic Boundary Conditions	30
Types Of Boundary Conditions	31
Parallel Flow Boundary Condition	31
Flow Boundary Condition	32
Water-Level Boundary Condition (Head)	33
Stagnation Point Boundary Condition	34
Typical Boundary Condition Examples	34
Boundary Condition Gotcha's	35
Bed Friction And Resistance To Flow	36
Defining Roughness	37
Modeling Turbulence	37
What Is Turbulence?	38
Specifying Turbulence	38
Results Listing Control	39
Full Results Listing	39
Summary Results Listing	40
Stopping The Simulation	40
Normal Run Completion	41
Immediately Stopping A Simulation	41
Any Comments?	42

Verifying The Simulation 43

Verification: A Process	43
Checking For Continuity	44
Continuity Check Lines	44
After The Simulation	44
Critical Check Points	44
Continuity Checks And Conservation Of Mass	44
Effects Of The Initial Conditions	45
Is The Model Realistically Simulating Wetting And Drying?	45

If Your Verification Fails	45
Wrong Choice Of Model	45
Geometry Problems	46
Boundary Condition Problems	46
Roughness Problems.....	47
Eddy Viscosity Problems	47
Influence On Verification	47

Interpretation Of Results 49

Overview	49
Understanding The Full Listing File	49
Model Information	50
Input Interpretation	51
Marsh Porosity Parameters	52
Element Connection Table	52
Echo Of Boundary Conditions	52
Nodal Specifications Or Initial Conditions.....	53
Steady State And Dynamic Simulation Progress And Statistics.....	54
Steady State And Dynamic Simulation Nodal Results	57
Understanding The Summary Listing File.....	57
Interpreting The Solution	58

Advanced Techniques 59

Using One Dimensional Elements.....	59
Using Basic One Dimensional Elements.....	59
Using One Dimensional Transition Elements	60
Using One Dimensional Junction Elements	60
Using One Dimensional Control Structures	62
Resuming A Stopped Simulation.....	64
Why Hotstart?	64
How To Hotstart	65
Common Causes For Hotstart Failure	67
Specifying Units.....	67
Customizing The Solution File.....	67
Throwing Out Initial Condition Contamination.....	67
Customizing The Full Results Listing File	68
When To Write To The Full Listing File	68
Including Startup Conditions.....	68
Including Your Input Data In The Full Listing File	69
Requesting Only A Summary Of Nodal Results	69
Plugging In Mesh Modifications	69
Modifying Nodes	70
Modifying Elements	70
Curving Element Edges.....	72
Changing Bottom Elevations	72
Resizing The Mesh.....	72
Changing An Element's Eddy Viscosity Tensor	72
Advanced Boundary Condition Techniques	73
Revising Boundary Conditions During A Simulation.....	73
Changing The Direction Of Flow	73
Changing The Speed Of The Current	74
Boundary Permeability (Reflection/Absorption).....	74

Varying Discharge During A Simulation (Rating Curve).....	74
Bendway Correction (Vorticity)	74
The Principle Of Bendway Correction	75
The Bendway Correction Solution Scheme In RMA2	75
How To Apply Bendway Correction	76
Fluid Properties	77
Fluid Density.....	77
Fluid Temperature.....	77
Automatic Friction Assignment	77
Automatic Turbulence Assignment	78
Wetting And Drying	78
Elemental Elimination	79
Marsh Porosity	80
Element Inflow And Outflow	83
Pumping And Discharging via Element Flow.....	83
Adding Rainfall And Evaporation	84
Compensating For The Rotation Of The Earth.....	85
Average Latitude Of The Mesh	85
Specifying Latitude By Material Type	85
Applying Wind Friction.....	85
Specify The Controlling Wind Formulation.....	86
Specify Wind Speed And Direction	86
Simulating With Storms	86
Storms In A Simulation.....	87
RMA2 Considerations	87
Defining Storm Events.....	88
Adjusting RMA2 Calculations	90

Run Control 91

Overview	91
Summary Of Run Control Data Cards.....	91
Input Variables	94
RMA2 Execution Job Sheet.....	99
RMA2 Data Cards	100
\$ L Card: Input/Output File Control	101
\$ M Card: Machine Identifier.....	103
BA Card: Boundary, Azimuth of Flow	104
BCC Card: Boundary Condition Control Parameters	105
BCN Card: Boundary Conditions by Node.....	106
BH Card: Boundary Head.....	107
BQ Card: Boundary Discharge	108
BRA Card: Boundary, Reflection/Absorption	110
BRC Card: Boundary Rating Curve.....	111
BS Card: Boundary Current Speed	112
BV Card: Boundary Inflow Vorticity	113
BW Card: Boundary, Wind Speed and Direction	114
BWC Card: Boundary Wind Formulation Control.....	115
BWS Card: Boundary, Wind Storm.....	120
CA Card: Special Calculation Variables.....	122
CO Card: Comments	123
DE Card: Wet/Dry by Elemental Elimination.....	124
DM Card: Wet/Dry by Marsh Porosity.....	126
END Card: End Card	129

EV Card: Turbulent Exchange Coefficients and Roughness.....	130
EX Card: Turbulent Exchange Coefficient, X-Velocity.....	132
EY Card: Turbulent Exchange Coefficient, Y-Velocity.....	133
FC Card: Flow Control Structures for 1D	134
FD Card: Fluid Density	136
FT Card: Water Temperature	137
G1 Card: Geometry, General Geometry Parameters.....	138
GC Card: Geometry, Continuity Check Line.....	139
GCL Card: Geometry, Continuity Check Line	140
GE Card: Geometry, Element Connection Table.....	141
GN Card: Geometry, Nodal Point Coordinates.....	142
GS Card: Geometry, Scale Factors.....	143
GT Card: Geometry, Element Material Types.....	144
GV Card: Geometry, Eddy Viscosity Tensor.....	145
GW Card: Geometry, One Dimensional Channel Width Attributes.....	146
GZ Card: Geometry, Nodal Point Elevation.....	147
HN Card: Roughness, Manning n-Value	148
IC Card: Initial Conditions	149
LA Card: Local Latitude.....	150
PE Card: Automatic Peclet Number Control of Eddy Viscosity	151
RA Card: Rainfall And Evaporation.....	153
RD Card: Automatic Roughness Coefficient Assignment by Depth.....	154
REV Card: Revise the Current Time Step.....	156
SI Card: System International Units	157
STO(P) Card: Stop the RMA2 Simulation	158
T1-T2 Cards: Job Title	159
T3 Card: Job Title	160
TI Card: Number of Iterations	161
TO Card: Time for Saving the Binary Results Solution File.....	162
TR Card: Full Results Listing Control	163
TRN Card: Summary Results Listing Control.....	164
TS Card: Timing For Binary Solution Output.....	165
TV Card: Iteration Control For Vorticity Calculations.....	166
TZ Card: Computation Time	167
VO Card: Vorticity (Bendway Correction).....	168
Performance Enhancements	169
Why Is RMA2 So Slow?.....	169
Computer Processor Speed	169
Temporary Files	169
Disk Performance	170
Reducing The Number Of Temporary Files	170
Effects Of An Oversized Buffer	170
Redimensioning RMA2.....	170
Common Problems	173
Common Problems and Remedies	173
Warning And Error Messages	179
Warning Messages.....	179
Error Messages	182

RMA2 File Formats	191
RMA2 Binary Output File Format.....	191
Alternate Dynamic Boundary Condition Input File Format	193
Utilities	195
FastTABS	195
Moving RMA2 Binary Solution Across Platforms	195
Subtracting Two RMA2 Solution Files	195
Merging RMA2 Solution Files	196
Averaging RMA2 Velocities	196
Obtaining Guideline Values Of Eddy Viscosity	196
Obtaining A Summary Listing After RMA2 Has Run	196
Technical Support	197
On-line Support	197
TABS Hotline	197
Glossary of Terms	199
Index	221
Notes	229

Overview

What Is RMA2

RMA2 is a two dimensional depth averaged finite element hydrodynamic numerical model. It computes water surface elevations and horizontal velocity components for subcritical, free-surface flow in two dimensional flow fields.

RMA2 computes a finite element solution of the Reynolds form of the Navier-Stokes equations for turbulent flows. Friction is calculated with the Manning's or Chezy equation, and eddy viscosity coefficients are used to define turbulence characteristics. Both steady and unsteady state (dynamic) problems can be analyzed.

Origin Of The Program

The original RMA2 was developed by Norton, King and Orlob (1973), of Water Resources Engineers, for the Walla Walla District, Corps of Engineers, and delivered in 1973. Further development, particularly of the marsh porosity option, was carried out by King and Roig at the University of California, Davis. Subsequent enhancements have been made by King and Norton, of Resource Management Associates (RMA), and by the Waterways Experiment Station (WES) Hydraulics Laboratory, culminating in the current version of the code supported in TABS-MD.

Applications For RMA2

The program has been applied to calculate water levels and flow distribution around islands; flow at bridges having one or more relief openings, in contracting and expanding reaches, into and out of off-channel hydropower plants, at river junctions, and into and out of pumping plant channels; circulation and transport in water bodies with wetlands; and general water levels and flow patterns in rivers, reservoirs, and estuaries.

Capabilities Of RMA2

RMA2 is a general purpose model designed for far-field problems in which vertical accelerations are negligible and velocity vectors generally point in the same

direction over the entire depth of the water column at any instant of time. It expects a vertically homogeneous fluid with a free surface.

RMA2 has these capabilities:

- Identify errors in the network.
- Accept either English or standard SI units. (Version 4.27 or higher)
- Restart (Hotstart) the simulation from a prior RMA2 run and continue.
- Simulate Wetting And Drying events.
 - Adjust for wetting and drying by element.
 - Account for Marsh Porosity wetting and drying (wetlands).
- Account for effects of the earth's rotation.
- Apply wind stress involving frontal (storm) passages.
- User selectable turbulent exchange coefficients, Manning's n-values, temperature, etc.
 - or user selectable equations for automatic dynamic assignment of Manning's n-value by depth (Version 4.28 or higher).
 - or user selectable Peclet number for automatic dynamic assignment of turbulent exchange coefficients (Version 4.28 or higher).
- Model up to 5 different types of flow control structures.
- Compute flow across continuity check lines.
- Provides for user defined computational guidelines; such as:
 - Wet/dry parameters
 - Iteration controls
 - Revisions within a time step
- Accepts a wide variety of boundary conditions.
 - Angle/velocity magnitude by node
 - Velocity components by node
 - Water surface elevations by node/line
 - Discharge by node/element/line
 - Tidal radiation by line
 - Discharge as a function of elevation by line
 - Wind speed and direction by node/element or element material type

Limitations Of RMA2

RMA2 operates under the hydrostatic assumption, meaning accelerations in the vertical direction are negligible. It is two dimensional in the horizontal plane. It is not intended to be used for near field problems where vortices, vibrations, or vertical accelerations are of primary interest. Vertically stratified flow effects are beyond the capabilities of RMA2.

RMA2 is a free-surface calculation model for subcritical flow problems. More complex flows where vertical variations of variables are important should be evaluated using a three dimensional model, such as RMA10.

Governing Equations

The generalized computer program RMA2 solves the depth-integrated equations of fluid mass and momentum conservation in two horizontal directions. The forms of the solved equations are

$$\begin{aligned}
 & h \frac{\partial u}{\partial t} + h u \frac{\partial u}{\partial x} + h v \frac{\partial u}{\partial y} - \frac{h}{\rho} \left(E_{xx} \frac{\partial^2 u}{\partial x^2} + E_{xy} \frac{\partial^2 u}{\partial y^2} \right) \\
 & + g h \left(\frac{\partial a}{\partial x} + \frac{\partial h}{\partial x} \right) + \frac{g u n^2}{(1.486 h^{1/6})^2} + (u^2 + v^2)^{1/2} \\
 & - z V_a^2 \cos y - 2 h w v \sin f = 0
 \end{aligned}
 \tag{1}$$

$$\begin{aligned}
 & h \frac{\partial v}{\partial t} + h u \frac{\partial v}{\partial x} + h v \frac{\partial v}{\partial y} - \frac{h}{\rho} \left(E_{yx} \frac{\partial^2 v}{\partial x^2} + E_{yy} \frac{\partial^2 v}{\partial y^2} \right) \\
 & + g h \left(\frac{\partial a}{\partial y} + \frac{\partial h}{\partial y} \right) + \frac{g v n^2}{(1.486 h^{1/6})^2} + (u^2 + v^2)^{1/2} \\
 & - z V_a^2 \sin y + 2 h w v \sin f = 0
 \end{aligned}
 \tag{2}$$

$$\frac{\partial h}{\partial t} + h \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) + u \frac{\partial h}{\partial x} + v \frac{\partial h}{\partial y} = 0
 \tag{3}$$

where

- h = Depth
- u, v = Velocities in the Cartesian directions
- x, y, t = Cartesian coordinates and time
- ρ = Density of fluid
- E = Eddy viscosity coefficient,
for xx = normal direction on x axis surface
for yy = normal direction on y axis surface
for xy and yx = shear direction on each surface
- g = Acceleration due to gravity
- a = Elevation of bottom
- n = Manning's roughness n -value
- 1.486 = Conversion from SI (metric) to non-SI units
- z = Empirical wind shear coefficient
- V_a = Wind speed
- y = Wind direction
- w = Rate of earth's angular rotation
- f = Local latitude

Equations 1, 2, and 3 are solved by the finite element method using the Galerkin Method of weighted residuals. The elements may be one dimensional lines, or two dimensional quadrilaterals or triangles, and may have curved (parabolic) sides. The shape functions are quadratic for velocity and linear for depth. Integration in space is performed by Gaussian integration. Derivatives in time are replaced by a nonlinear finite difference approximation. Variables are assumed to vary over each time interval in the form

$$f(t) = f(0) + at + bt^c \quad t_0 \leq t < t_0 + \Delta t \quad (4)$$

which is differentiated with respect to time, and cast in finite difference form. Letters a , b , and c are constants.



Note: It has been found by experiment that the best value for c is 1.5 (Norton and King 1977).

The solution is fully implicit and the set of simultaneous equations is solved by Newton-Raphson non linear iteration. The computer code executes the solution by means of a front-type solver, which assembles a portion of the matrix and solves it before assembling the next portion of the matrix.

RMA2 is based on the earlier versions (Norton and King 1977) but differs in several ways.

It is formulated in terms of velocity (\mathbf{v}) instead of unit discharge (\mathbf{vh}), which improves some aspects of the code's behavior.

- Employs new numerical solution algorithms.
- Permits wetting and drying of areas within the grid.
- Permits wetlands to be simulated as either totally wet/dry or as gradually changing wet/dry states.
- Permits specification of turbulent coefficients in directions other than along the x and z axes.
- Accommodates the specifications of hydraulic control structures in the network.
- Permits the use of automatic assignment of friction and turbulent coefficients.
- Permits input in either English or System International units.
- Permits vorticity.

Element Types Supported

RMA2 is capable of supporting different types of elements within the same computational finite element mesh. The types of elements fit into three basic categories:

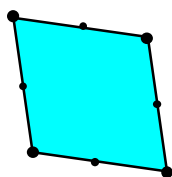
- Two Dimensional Elements
- One Dimensional Elements
- Special Elements



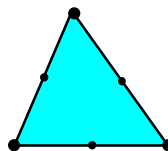
Note: Element edges may be either straight or curved.

Two Dimensional Elements

Two dimensional elements are the customary type used with RMA2 and may be either triangular or rectangular in shape. A two dimensional element possesses a length and a width, determined by the positions of the corner nodes which define the element. The depth at any location within a two dimensional element is obtained by interpolating among the depths of the corner nodes which define the element.



Rectangular Element



Triangular Element

One Dimensional Elements

A one dimensional element is a simplified element which is composed of two corner nodes and one midside node. The Finite Element Governing Equations for one dimensional elements are based on a trapezoidal cross section with side slopes, and an off channel storage area. The depth at any location along a one dimensional element is obtained by interpolating between the depths of the two corner nodes which define the element. One dimensional element characteristics are defined using GN and GW cards with the N option. See “Using One Dimensional Elements” on page 59 for more information.

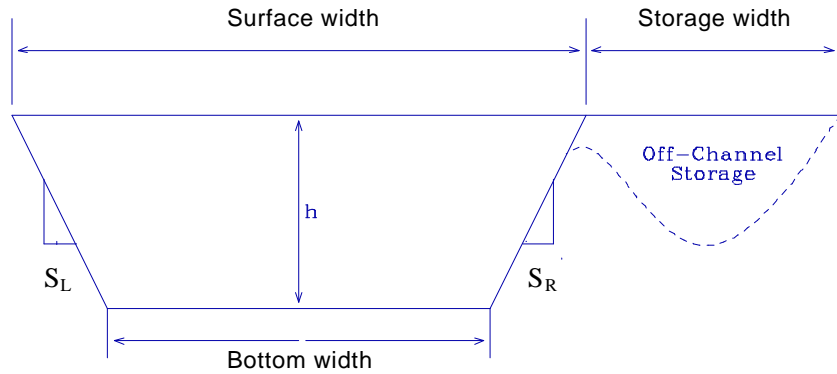


FastTABS Note: As of this writing, in order to allow FastTABS to recognize one dimensional element widths and side slopes, you must use a GN card with the N option to specify the x and y coordinates and the bottom elevation, and a GW card with the N option to specify the surface width, side storage width, and left and right side slopes.

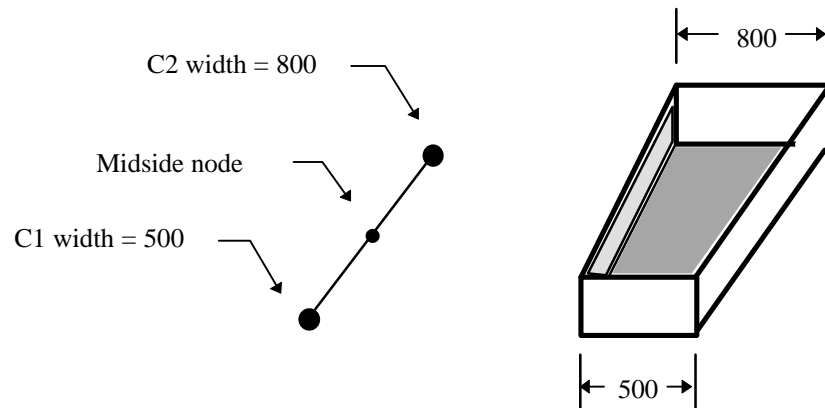
The Basic One Dimensional Element

The basic one dimensional element is composed of two corner nodes and one midside node, and may be either straight or curved.

The numerical model’s Governing Equations for one dimensional elements are based on a trapezoidal cross-section with side slopes and off-channel storage. To describe the trapezoidal cross-section, you must assign for each one dimensional corner node a surface width, a left and right slope (S_L and S_R), and an off channel storage width. If the values of S_L , S_R , and the off channel storage width are zero, the trapezoidal shape reduces to a rectangle.



A basic one dimensional element can have a different width at each corner node. A basic straight sided element with zero side slopes, but different width assignments at each corner (figure on left) will have a shape that looks like the figure on the right:



Special Elements

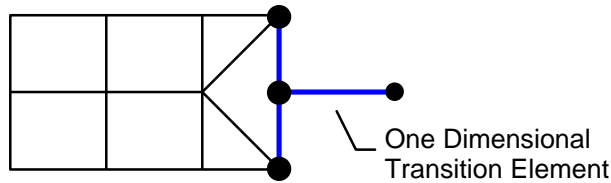
Some one dimensional elements serve special purposes. These elements fall into three categories:

- Transition Elements
- Junction Elements
- Control Structure Elements

See “Using One Dimensional Elements” on page 59 for more information on these special elements.

Transition Elements

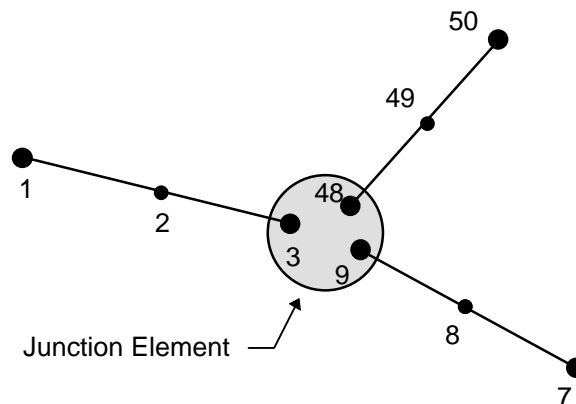
A transition element is required anytime you want to use a one dimensional element in your mesh. This type of one dimensional element makes the transition between the two dimensional elements and the one dimensional elements.



For additional information on Transition elements, see “Using One Dimensional Transition Elements” on page 60, and the GFGEN reference manual.

Junction Elements

A Junction element is a special one dimensional element used to describe the proper characteristics where *three or more* one dimensional elements intersect. Junction elements are defined using IMAT values of 901, 902, or 903. The junction element *is the point* where the other one dimensional elements connect. There are as many nodes defining this junction element as there are one dimensional elements connecting to it.



GE elem# 3 9 48 0 0 0 0 0 imat#

For example, if there are 3 one dimensional elements connecting at a point, then that point is the junction element, and it contains 3 nodes.



Note: The functional limit of the model on the number of elements entering a junction is 8, as dictated by the number of nodes in an element.

For additional information on Junction elements, see the GFGEN reference manual.

Types Of Junction Elements

There are three categories of Junction elements:

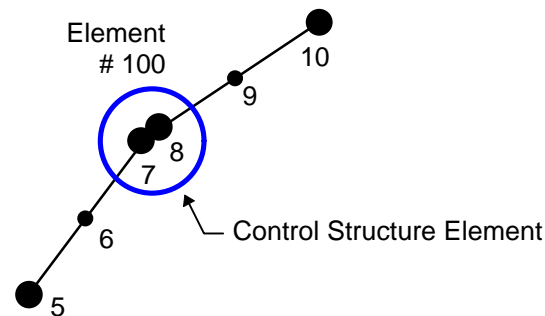
- Water Surface Junction (IMAT = 901)
All water levels match at the junction.
- Total Head Junction (IMAT = 902)
All total energy heads match at the junction.
- Momentum Junction (IMAT = 903)
Momentum is conserved in the primary channel. The first 2 nodes in

the junction element define the primary channel within which momentum will be conserved. Water levels for junction ends of the remaining elements all are set to the average water levels of these first two nodes.

Control Structure Elements

A Control Structure element is used to simulate obstructions in the flow path, such as weirs, dams, flood gates, etc.

A Control Structure element is a single point which contains two nodes and has an IMAT value ≥ 904 . The order of the node numbering at a Control Structure element should be that the side with higher elevation comes first, then the side with the lower elevation. This is generally the “upstream” side of the structure followed by the “downstream” side.



GE 100 7 8 0 0 0 0 0 904

For additional information on Control Structure elements, see the GFGEN reference manual.

Curved Element Edges

Although it is no longer necessary to prescribe curved external boundaries to prevent ‘leaking’ (releases prior to RMA2 version 4.20), they may be used to achieve aesthetics when viewing the mesh, add length without additional resolution, and to aid in mass conservation for transport applications.

An element edge may be *curved* by moving the (x, y) location of the midside node. See “Curving Element Edges” on page 72 for more information.



Note: It is not advised to curve interior element edges.



Tip: Curving is particularly useful for one dimensional elements to achieve the meandering/snaking river effect.

What’s New And What’s Gone

During the process of writing this manual, the RMA2 source code has been reviewed many times. As a result, many code errors have been eliminated, and some input procedures have been added or changed in an attempt to make the use of

RMA2 more straightforward. For more information on many of these changes, see “R” on page 91.

Ch- Ch- Ch- Ch- Changes

BQ Card Changes

The BQ card with the L option now provides the capability to specify the flow distribution weighted by water depth across a channel (QXP variable).

G1 Card Changes

The G1 card is no longer supported. Use instead the

- GS card for changing x, y, and/or z scale factors.
- LA card to specify local latitude.
- BCC card to specify the counter for the number of BCN cards (NBX variable).

GC Card Changes

Use of the GC card will result in a warning stating that the GCL card is the preferred method to define continuity check lines. The continuity line number is now defined at the start of the GCL card, and not by the card order, as is the case with GC cards.

GY Card Changes

The GY card used to specify bottom elevations has been renamed GZ Card.

TR Card Changes

The on/off switch for the summary results listing file (ISPRT variable) was eliminated from the TR card. Use the \$L card for this feature.

Additions

Bendway Correction (Vorticity) Added

An improved method of calculating velocities around curved channels has been incorporated into RMA2. New data cards were added to accomplish this, and are the BV card, TS card, TV card, and VO card. See “Bendway Correction (Vorticity)” on page 74 for more information.

GCL Card Added

The GCL card is a replacement for the GC card used to define continuity line numbers. The continuity line number is now defined at the start of the GCL card, and not by the card order, as is the case with GC cards.

GZ Card Added

The GZ card directly replaces the GY card for specifying bottom elevations. The GY card was simply renamed.

Rainfall And Evaporation Added

A simple means to include rainfall and evaporation in a simulation was added to RMA2. The RA card is used to supply the parameters for this new method. See “Adding Rainfall And Evaporation” on page 84 for more details.

Deletions

\$F Card Deleted

The \$F card for fixed data field format is no longer supported.

System Requirements

Because RMA2 is written in standard FORTRAN 77 code, it can be compiled and executed on many different types of computer systems. System requirements such as RAM and disk space will differ depending upon the size of your project. The size of your project may require you to choose to use one type of system over another, or require an upgrade of your current system. In general, you can use the table below as a guide.

Max Number Of Elements	Required RAM (Mbytes)
2500	4
5000	8
8000	16
15000	32



Note: The RMA2 source code must be edited to re-dimension arrays pertaining to the number of elements, nodes, etc., if the current dimensions are not adequate for your project.

In addition to RAM, the processing power, or speed of your system is also an important consideration when using RMA2. RMA2 can require a large amount of computational power during a simulation, and can take a very long time to complete the run. To be most efficient, try to use a system with a good, powerful processor.

In some cases, machine precision may become an issue. Different types of computer systems can have different word lengths, and this word length can affect precision. If you suspect RMA2 will encounter very large or very small numbers during a simulation, be aware that machines with a smaller word length, such as PC's, may not perform to your expectations.

For additional information, see “Performance Enhancements” on page 169.

Personal Computer Systems

The most commonly used personal computer systems on which RMA2 is run are the “IBM compatible” PC, and the Macintosh. Remember that the size of your project dictates the amount of RAM your system will actually need.

PC's (IBM Compatible)

The minimum requirement for a PC is a 386 or better CPU with 4 Mbytes of RAM, and 20 Mbytes of free disk space, and the DOS extender program DOS4GW.EXE from Rational Systems™.

If you do not have enough physical memory to run RMA2, and you have a windowing program such as Microsoft™ Windows™, which allows the use of DOS virtual machines or DOS windows, you may be able to run RMA2 in this environment. You need to be sure under this environment that the size allocated for *virtual memory* will provide you with enough total RAM to execute RMA2. Also, be aware that RMA2 may run much slower with this configuration than if your system actually had enough physical RAM.

Macintosh

Requires a 68030 or better CPU, 4 Mbytes of RAM, and 20 Mbytes of free disk space.

Mini Computer Systems (Workstations)

Workstations are a good choice for running RMA2. They provide a stable platform without the computational costs associated with many mainframe computer systems. Today's workstations are normally shipped with at least 32 Mbytes of memory, and will usually not have a problem running RMA2. Be sure that you have at least 20 Mbytes of free disk space for installation and minimal execution.

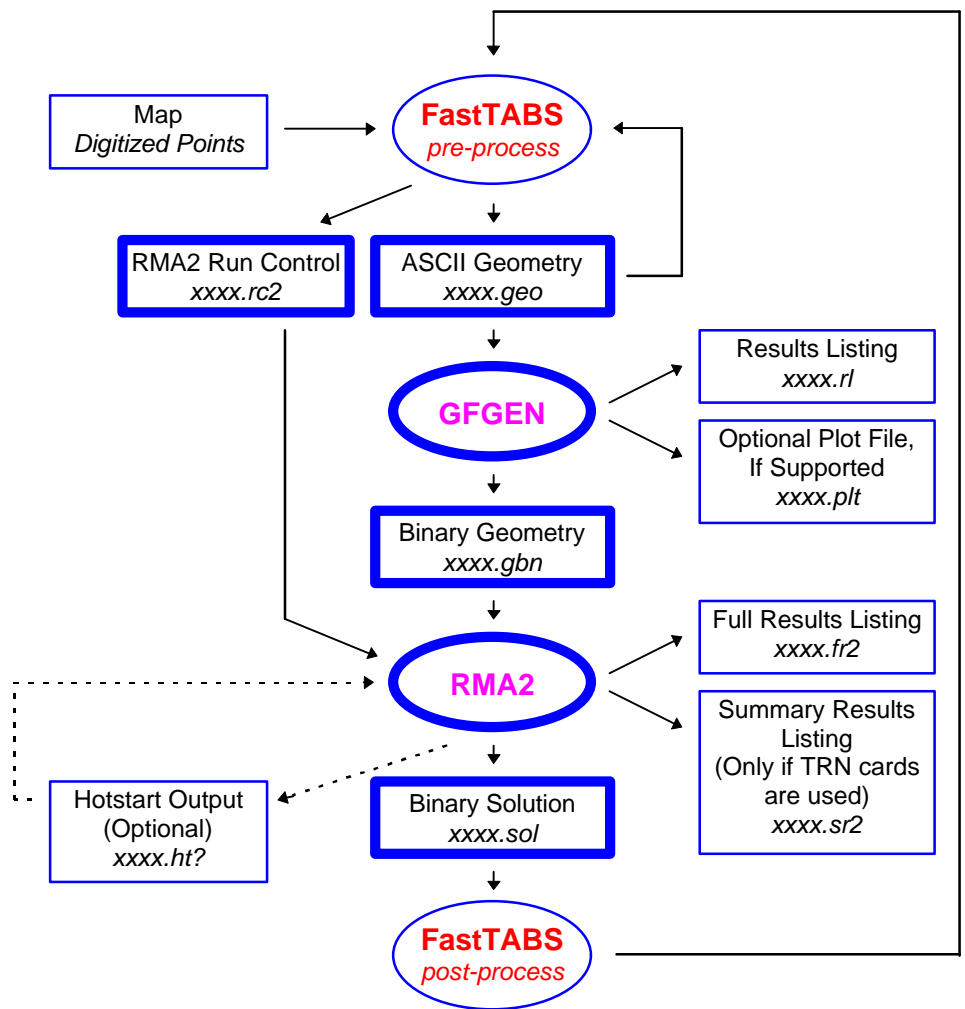
Mainframe/Supercomputer Systems

If you have access to one, a mainframe, or supercomputer may be very useful when running RMA2. Many are faster and use more precision than do some workstations. Be sure, however, to have enough available disk space, at least 20 Mbytes, allotted to you for installation and storing RMA2 run output.

Using RMA2

The RMA2 Modeling Process

The following flow chart illustrates the RMA2 modeling process. Items with bold borders are required, others are optional.



When RMA2 is used for a dynamic (unsteady state) simulation, it first must solve a steady state problem. The results from this steady state case are used to start the dynamic simulation. Unless specified otherwise, RMA2 will continue directly from the steady state solution and begin the dynamic simulation.

Modes Of Operation

Typically RMA2 is set to batch mode for mainframe computer environments and to interactive mode in PC and workstation environments.

RMA2 may be executed in either *batch mode* or in *interactive mode*. The mode is determined by the value of the IBATCH variable within the program. You can change the mode only if you have the source code and a FORTRAN compiler.

Running In Interactive Mode

To run interactively, execute RMA2 directly and answer the series of questions referring to input and output file names. For instructions, enter a ‘?’.

The figure below illustrates the startup procedure for RMA2 in interactive mode. Note that the question regarding a particular file will only be asked if that file is active on the \$L card. You may choose to null or prevent that file from being opened by entering a ‘**null**’ in response to the file name. For information on RMA2 files, see “Data Files” on page 19.

RMA2 VERSION 4.30 1-D AND 2-D
CAPABILITY. LAST MOD DATE 09-01-1995

<---- Version identifier
Last Modified

THIS EXECUTABLE IS DIMENSIONED AS FOLLOWS:

MAX NO. OF NODES	1000
MAX NO. OF ELEMENTS	400
MAX NO. OF EQUATIONS	2000
MAX FRONT WIDTH	120
MAX NO. OF CONTINUITY CHECK LINES	15
MAX NO. OF NODES PER CONTINUITY LINE	40
MAX BUFFER SIZE	4000
MAX PRINT-SUMMARY BUFFER	25000

<---- Dimensions

===== RMA2 VERSION 4.30 1-D AND 2-D =====
===== CAPABILITY. LAST MOD DATE 09-01-1995 =====

===== TABS-MD FE HYDRODYNAMIC MODEL =====
///// Original Author: Ian P. King of RMA /////
///// Modified and Maintained by WES-HL /////

Enter a --> ? to receive a response menu

<---- File Prompting

ENTER RUN CONTROL INPUT FILE NAME
ehibase.n

ENTER FULL PRINT OUTPUT FILE NAME
ehobase.fp

RMA2 VERSION 4.30 READING INPUT DATA ... UNIT= 2

NO BANNERS ON INPUT CONTROL FILE ... REWIND

T1 RMA2 4.3 INPUT DYNAMIC-40 HRS Run #4 9-9-91 (208 ELEM-629 NODES)

T2 Cold start with revision to hr 40 9-06-1995

T3 GENERIC ESTUARY- BASE IMAT1-2 (150-.025) IMAT3 (100-.01) MARSH POROSITY

\$M 5

\$L 0 62 60 64 5 3 59

ENTER ALTERNATE B-C INPUT FILE
null

<---- Null a file

NOTE: You just turned off Alternate BC file
ENTER SPECIAL PRINT OUTPUT FILE
ehobase.sp

ENTER INPUT GEOMETRY FILE (binary)
egobase.ibin

ENTER OUTPUT RESTART/HOTSTART FILE (binary)
ehobase.hot

ENTER FINAL RMA-2 RESULTS FILE (binary)
ehobase.bin

--> Just Read GFGEN.. nodes & elements=629, 208

Interactive RMA2 Session.

Running In Batch Mode

Batch mode is typically used on a large mainframe computer system, such as the Cray C-90 at the Waterways Experiment Station. On the WES Cray computers, batch mode execution of RMA2 for valid TABS-MD users is accessible by typing **proclv** and answering the series of questions referring to input and output file names.

Why Use Batch Mode

If you have access to the Cray computers at WES, or another mainframe computer containing the TABS programs, you may wish to execute RMA2 in batch mode — especially if you have a large problem. Generally, you will have access to more CPU time and more system RAM when executing in batch mode. The table below shows the differences between available system resources when executing in batch mode vs. interactive mode on the WES Cray computers.

Execution Mode	Max CPU Time	Max System RAM
Batch Mode	24 Hours	192 Mbytes
Interactive Mode	10 Minutes	2 Mbytes

Another advantage of batch mode is that it does not tie up your terminal. Once the job has been submitted to the system, your terminal is free and available for you to perform other tasks.

Guidelines For Obtaining A Good Solution

All aspects of the geometry and the numerical model simulation must run in harmony. In addition to the geometry, the RMA2 run control (boundary condition) file must contain the proper information if the simulation is to be successful. A graphical user interface such as FastTABS will help you build a run control file, but it is still recommended that you examine that file and double check the run control selections.



Note: The primary requirement for a successful numerical model is preparing a good mesh; developing it with the following recommended guidelines in mind. These guidelines are briefly described below and can be found in detail in the GFGEN geometry manual.

Maintain Good Element Properties

Some good element properties:

- Maintain a length to width ratio of less than one to ten.
- Restrict element shapes to undistorted triangles or rectangles.
- Create elements with corner angles greater than 10 degrees.
- All bathymetric elevations should lie in one plane.
- Maintain longitudinal element edge depth changes of less than 20%.

Maintain Good Mesh Properties

- A well constructed mesh must first have good element properties.
- The overall bathymetric contours should be smooth.
- Wetting and drying studies need the element edges to lie on bathymetric contours.
- Ideally, any *boundary break angle* should not exceed 10 degrees.
- Neighboring elements should not differ in size by more than 50%.
- Use adequate resolution to model the features of the prototype plan field.

Match Resolution With The Situation

Before designing the mesh, consideration should be given to the purpose of the study. There is more to mesh design than having well formed elements and good mesh properties. The resolution requirements for a hydrodynamic study must also consider the end use of the solution. Resolution is not only an issue of distance from the study area, but an issue of supporting post-hydrodynamic purposes, such as the sediment transport model or the WES ship simulator model. The table below illustrates the minimum resolution requirements that are necessary to adequately define the hydrodynamics for various circumstances.

Interested in...	Type of Element	Minimal Degree of Resolution
General circulation	2D	Coarse
Eddy patterns	2D	Two elements spanning the prototype diameter of the eddy
Minimum flow exchange	2D	Two elements wide
Well-confined channel away from study area	1D and/or 2D	One element wide
Navigation channel for ship simulator	2D	Four elements laterally across the channel
Turn approaches for ship simulator	2D	One element per ship length in the longitudinal direction of flow
Turning basins for ship simulator	2D	Two elements per ship length
Sedimentation Issues	2D	Fine resolution is required to pick up deposition/erosion patterns
Non-stratified salinity intrusion in a channel	2D	Three elements wide

Pre-RMA2 Simulation Check List

After the mesh has been constructed, the following check list can be helpful. Be sure to:

- Renumber the mesh with a graphical user interface such as FastTABS.
- Run GFGEN to reorder the mesh and save a binary geometry file.
- Verify that the machine identifier on the \$M card is correct for your computer system.

- Set the appropriate input/output switches for file control on the \$L card.
- Set the timing and iteration control on the TI card and TZ card.
- If the simulation is likely to cause wetting and drying, a DE card and/or DM card will be required.
- Evaluate your selection of roughness and turbulent exchange coefficients.
- Make sure that all nodes in the mesh have been assigned roughness and turbulent exchange values.
- Check that the Coldstart initial condition on the IC card is equal or close to the water surface elevation boundary condition on the BH card.
- Evaluate the location and type of boundary conditions.
- Check the inflow angle of all boundary conditions.

Persistent Divergence

If convergence problems persists after you have checked the mesh characteristics and pre-simulation check list, you should examine the solution to determine where the simulation *first* appears to begin diverging. With a graphical user interface such as FastTABS, you can view the various time steps, or iterations, from the run to find the location in the mesh, and the time, where the problem begins.

Depending on the circumstances, the model may diverge very slowly or very suddenly. The rate of convergence or divergence can be monitored by examining the maximum change category of the convergence parameters found in the full results listing file (also echoed to the screen or logfile). Slow divergence usually indicates a run parameter problem, while sudden divergence typically indicates boundary condition errors, wetting and drying problems, or super-critical flow.



Note: A momentary transient effect may be mis-interpreted as divergence whenever there has been an update in the run control parameters, modification due to wetting and drying, or a change of boundary conditions (a new time step or REV card). This effect is normal, and a well constructed model should iterate to a converged solution for a given time step.



Tip: If you are unable to examine the solution at the time it first begins to diverge, try re-running the simulation with artificially higher values of eddy viscosity and/or force the model to prematurely stop just prior to divergence.

Basic Operation

Data Files

RMA2 may read and write several files during a simulation. The number and type of files depends upon choices you make about how the simulation will run and what type of information you want to see in the results.

The \$L card is used to specify the files that will be utilized by RMA2. The types of files RMA2 uses consist of

- RMA2 run control files (normally named with the extension **.rc2** or **.bc**)
- GFGEN binary geometry data
- Hotstart input and output
- Listings of results
- Solution files in binary form containing the RMA2 numerical results.

For a normal run, you should specify as input a run control file and the binary geometry file from GFGEN, and as output a full results listing and a solution file. See “The RMA2 Modeling Process” on page 13 for additional information.

Run Control Files

There are many options available to you when running an RMA2 simulation. The run control file (a.k.a. boundary condition file) is what you use to tell RMA2 how to run the simulation. Every action taken by RMA2 is defined or modified in the run control file.

In addition to the run control file, you may specify an alternate file for dynamic boundary condition data. This additional file is used in conjunction with the primary run control file. The \$L card is used to specify an alternate boundary condition file.

Once control has been transferred to the alternate boundary condition file, the primary run control file cannot be accessed again.



Note: It is recommended that you use the primary run control file for all RMA2 input. The use of an alternate dynamic boundary condition file is available but not supported.

A typical run control file is shown below.

```

T1  EXAMPLE: STEADY STATE
T2  Run control file
T3  NOYO : PLAN 1
$L  00      0      60      64      0      3      0
SI  0
TR   0      -1      1      0      1
G1  0.0      1.0      1.0      00
GC   4      631      633      636      639
GC   8      1       7      12      17      22      27      32      37
TZ   0.0      0.0      100      1      0
TI   3      0       0.0001      0.0001
FT  17.0
IC  15.0      0.2      0.25
EV   1      100.00      100.00      100.00      00.00      0.015000
EV   2      100.00      100.00      100.00      00.00      0.050000
EV   3      50.00      50.00      50.00      00.00      0.030000
BQL  1      1500.0      3.14
BHL  2      16.0
END
STOP

```

It is possible to code the entire mesh in the RMA2 run control file. However, this is highly discouraged. The ability to code mesh geometry for RMA2 is intended for making minor geometry modifications when testing a proposed change to the mesh.

Geometry File

The mesh geometry which RMA2 will use is normally defined in a binary file produced by the Geometry File Generation program, GFGEN. This mesh geometry file consists of the nodes and elements that define the size, shape, and bathymetry of the study area.

RMA2 will not read a text, or ASCII, type file as input for the mesh geometry. If you do not have a binary geometry file, and you do not want to code the mesh geometry in the RMA2 run control file, you must run GFGEN to obtain the geometry file before running RMA2. The \$L card is used to include the geometry file in the simulation run.

Hotstart File

Hotstarting may be desired when you have a limit on run time for a simulation, or you only want to retain certain time intervals of the solution. A Hotstart file is used to preserve the critical information (derivatives, etc.) at the end of a simulation in order for the run to be restarted and continued at a later time.

A Hotstart file can be both an output file and an input file. The Hotstart output from run 1, for example, is typically used as the Hotstart input for run 2. When a Hotstart file is used as input, it defines the initial conditions for the new simulation.

For more information on Hotstarting, see the sections entitled “Stopping The Simulation” on page 40 and “Resuming A Stopped Simulation” on page 64.

Results Listing Files

The listing files can contain a plethora of information pertaining to the simulation results. Upon normal completion of an RMA2 simulation run, a results listing file, or files, may be written, depending upon the settings of the parameters that pertain to results listing files on the \$L card.

For details on creating results listing files, see the sections entitled “Results Listing Control” on page 39 and “Customizing The Full Results Listing File” on page 68.

Full Results Listing

If a full listing of results is desired, use the \$L card to specify a full listing file. The TR card is used to specify what types of information will be written in the file. The full results listing file may contain

- Echo of the run control data
- Geometry information
- Wetting and Drying/Marsh Porosity information
- Element information
- Nodal specifications
- Front width size and total equations
- Convergence parameters and active nodal statistics for each iteration
- Nodal velocity, depth, and elevation results
- Continuity check lines
- Boundary condition updates

See “Understanding The Full Listing File” on page 49 for details.

Summary Results Listing

When you only need a summary of the simulation results at specific nodes, you can request a summary listing file using the \$L card. The nodes to be included in the summary listing file are specified using the TRN card. The summary results listing file contains

- A table of the steady state solutions of x and y velocity components, depth, etc., for each node listed on the TRN cards
- A dynamic hydrograph of the above information for each node listed on TRN cards



Tip: Information in the summary listing can be imported into a spreadsheet program for plotting and further analysis.

For more information on the summary results listing, see “Understanding The Summary Listing File” on page 57.

Solution File

RMA2 will write the final solutions from its calculations to a solution file. The solution file is a binary file that, upon normal completion of the RMA2 simulation run, contains the results of computations for all time steps defined in the run control file (see “Run Control Files” on page 19). The data in the solution file can be graphically analyzed using a compatible post-processor, such as the FastTABS interface.

The solution file contains the following types of information for each node in the mesh:

- x component of velocity
- y component of velocity

- Depth
- Water surface elevation
- A flag to mark as wet or dry (NDRY)

The solution file also contains the material type number for each element, which is written as negative if the element is dry.

Vorticity Solution File

The vorticity solution file is a look-alike of an RMA4 solution file, which contains the value of vorticity at every node. Streamwise vorticity (sec^{-1}) at a node is equal to twice the rotational velocity of the fluid about the axis in the streamwise direction of flow. The vorticity solution file also contains the material type number for each element, which is written as negative if the element is dry.

The existence of the vorticity solution file is controlled on the both \$L card and the TS card together. The \$L card tells RMA2 to ask you for a vorticity solution file name, and the TS card controls whether the vorticity solution is actually written to the file, and when. For additional information on vorticity, see “Bendway Correction (Vorticity)” on page 74.

Using Titles

The ability to add Titles in the RMA2 run control input file provides a means to describe the data which is being modeled. Titles are specified using the T1 and T2 cards, and a T3 card.



Note: A Title card must be the *first* card in the run control file, otherwise RMA2 will not recognize the file as valid input.

Including Title Information

Enter the Title and descriptive information about your data on T1, T2, and T3 cards. You may use as many T1 and T2 cards as you wish, and card order is unimportant. Be sure to end your set of Title cards with a T3 card.

The Last Title Card

The last Title card is the T3 card. Only one T3 card is allowed and it must be the very *last* title card. RMA2 reads the ‘3’ to mean the *end* of the Title cards.



Tip: The information on the T3 card is retained by RMA2 and is written into the binary solution file header. Use the T3 card to your advantage. Supply information which will allow your solution file to be more easily identified in the future.

What Kind Of Computer Do You Have?

Hey, What Do You Care?

Because different computer systems may store and retrieve information in different ways, RMA2 needs to know the type of system on which it is running so it can properly transfer information on the system.

RMA2 solution files and buffer files are written in a binary form. Binary files are strictly associated with the type of system upon which they are created. Word size and record length may be different from one system to the next.



Note: The computer identification is vital only if there is insufficient memory allotted during execution of RMA2 which would require it to write temporary buffer files to solve the large matrix.

So What Do You Want Me To Do About It?

Tell RMA2 what type of system on which you will be running by providing a value for the machine identifier with the \$M card. The type of computer you specify determines how temporary buffer files will be written and read.

The \$M card is necessary if your system does not have enough memory available for the simulation. In this case, RMA2 will write temporary buffer files to your disk. See the chapter entitled “Performance Enhancements” on page 169 for additional information on temporary buffer files.



Example: If you are running RMA2 on a DOS based PC, the machine identifier value should be 1.

```
.  
CO      Running on a DOS PC  
$M      1  
.br/>.
```

Specifying Initial Conditions

To begin a simulation, RMA2 must have some values to use as initial guesses for the equations it will be solving. RMA2 must know the conditions for all nodes in the mesh at the instant in time depicted when the simulation begins.



Warning: In the early stages of a simulation, the solution may be contaminated by the inaccuracy of the initial conditions. Typically, for a dynamic estuarine simulation, you would ignore at least the first 24 hours of the solution.

One dimensional nodes should have a minimal depth and a nominal velocity assigned with the IC card.

Coldstart Initial Conditions

For a Coldstart run, RMA2 begins with a global, flat, water surface elevation as specified on the IC card. Two dimensional velocities are zero, and one dimensional velocities are a nominal amount, also specified on the IC card. The depth at each node is calculated as the difference between *initial water surface elevation* specified on the IC card, and the bed elevations from the mesh geometry.

When coldstarting with large bed slopes (such as 1 on 10), it may be helpful from a convergence standpoint to specify the initial condition water surface elevation higher than the highest bed elevation, and the minimum water depth to be greater than the critical depth for wetting and drying (DSET on the DE card); i.e., everything is wet. This will generally allow the model to more easily arrive at a solution. Because this initial water surface elevation may be artificially high, you can use either the REV card or Hotstart to gradually lower the water surface to the realistic value.

Hotstart Initial Conditions

Data on the IC card is ignored for a Hotstart run.

Any initial conditions supplied by the user are ignored when Hotstarting because they are read from a Hotstart file generated from a previous RMA2 simulation. The \$L card is used to indicate the use of Hotstart files.

See “Resuming A Stopped Simulation” on page 64 for more information on Hotstarting RMA2.

Iteration Control

It is nearly impossible to predict the exact number of iterations that will be required to solve a particular problem. Therefore, it is recommended to specify the *maximum* number of iterations to attempt per time step, in combination with a satisfactory depth convergence criterion. This maximum number of iterations attempted for each time step, and the depth convergence criterion are specified using the TI card.

What Is An Iteration?

Webster’s defines *iterative* in the mathematical sense as

Of, relating to, or being a computational procedure to produce a desired result by replication of a series of operations that successively better approximates the desired result.

Basically, the process of *iteration* is performed by making an initial guess for the values of the variables in the equation to be solved, computing a solution, replacing that solution back into the equation, and repeating the computation. When the difference between the results from two successive computations is less than some predetermined amount, the solution is said to have *converged*, and the problem is considered solved.

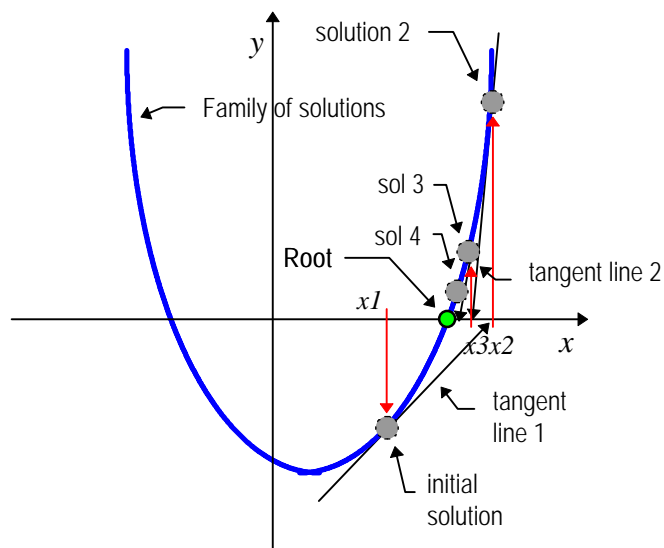
How RMA2 Finds A Solution

RMA2 uses the Newton-Raphson convergence scheme to obtain a solution. To see how this method works, see the figure below. The example is in the form of the

family of solutions to a typical quadratic equation. The idea is to find a solution to the equation as close to the root as possible.

On the x axis, x_1 is the initial guess at a value which will yield the optimum solution. The solution with x_1 is the point marked “initial solution”. A line (tangent line 1) which is tangent to the curve at this point is computed. The place where this line crosses the x axis becomes x_2 ; the second guess used to solve the problem.

A new solution is calculated from x_2 , and another tangent line (tangent line 2) is computed. The point where this tangent line crosses the x axis becomes the next guess, x_3 . And so on, until the difference in value along the x axis, between two successive solutions, becomes less than the a pre-defined convergence criterion. At this point, the solution has converged.



where

x_1 = the initial guess at the solution

x_2 = the next guess, which is the solution obtained from x_1

x_3 = the next guess, which is the solution obtained from x_2

The problem is considered solved when the difference between two solutions is less than the predetermined minimum amount.



Note: If the first estimate is very far from the solution, the iterative technique may diverge.

For additional information, see mathematical texts discussing the Newton-Raphson method.

Regulating The Number Of Iterations

In RMA2, the Newton-Raphson iterative method of successive approximations is used to solve a system of simultaneous equations. The procedure requires an initial approximation, called *Initial Condition*, to start the computations. Convergence from that first estimate to the final solution usually takes three to five iterations. Of

course, the better the first estimate approximates the solution, the fewer the number of iterations required for convergence. If the first estimate is very far from the solution, the iterative technique may diverge.

RMA2 provides two means by which to control the number of iterations that are performed per time step. They are

- Providing A Convergence Criterion
- Directly Specifying The Maximum Number Of Iterations

No more iterations are performed when either of these situations are satisfied.

Providing A Convergence Criterion

The criterion used by RMA2 to test for convergence is the maximum change in calculated depth of all nodes in the mesh from one iteration to the next. The listing below shows a representative sample of a convergence parameter section from the output which RMA2 will produce at the end of each iteration. The Depth is the third *degree of freedom* in the RMA2 equations.

The circled value in the MAX CHG column in the figure is compared to the convergence criterion you supply on the TI card. When this calculated value is less than your convergence criterion, then, unless you are using REV cards, the calculations for this time step are finished and RMA2 advances to the next time step. Otherwise, if you have REV cards, calculations with the current set of data are finished, and RMA2 advances to the next revision data set for the time step.

0.0813 in the MAX CHG column is compared to the Depth Convergence Criterion you specified on the TI card. If it is smaller, RMA2 advances to the next data set or time step.

RESULTS AT THE END OF 1 TIME STEPS...

TOTAL TIME = 0.0 HOURS... ITERATION = 10

CONVERGENCE PARAMETERS

DF	AVG CHG	MAX CHG	LOCATION
1	0.0175	0.1885	384 X-VEL
2	0.0641	-0.2638	153 Y-VEL
3	0.0190	0.0813	563 DEPTH

The value for the depth convergence criterion is supplied on the TI card. The value typically varies between 0.005 to 0.0001 for steady state, and between 0.05 to 0.001 for dynamic runs. A more stringent criterion is required for true steady state simulations, and/or wetting and drying simulations. RMA2 will continue the Newton-Raphson iteration process until either the maximum number of iterations, or the depth convergence criterion, has been satisfied.



Note: Specifying a depth convergence value of **zero** is certain to cause RMA2 to compute for the maximum number of iterations you specified because a difference in solutions of absolutely zero is highly improbable. However, this is valid, and you may want to do this in some instances.

Directly Specifying The Maximum Number Of Iterations

The maximum number of iterations for both steady state and dynamic runs should be large enough to allow the model to sufficiently converge with the given conditions. The required number of iterations must be larger for complex hydrodynamic studies, such as simulations where wetting and drying occur and the

hydrodynamics changes from one iteration to the next. The TI card is used to provide these values to RMA2.



Warning: If the problem at a given time step does not converge to a good solution in the allotted number of iterations, the solutions for future time steps may suffer as a consequence.

Now, What Do I Really Do?

A General Approach To Specifying Iterations

To eliminate extensive use of CPU resources, you may wish to limit the number of iterations performed per time step. You want to use the smallest number of iterations possible, and still solve your problem sufficiently. A trial and error technique may be necessary if you wish to optimize this number.



Tip: Experience has shown that, for *simpler* problems (no wetting and drying, etc.) with no sudden changes in boundary conditions between time steps, the number of iterations per dynamic time step can usually be reduced to two, and the dynamic depth convergence criterion can be zero.

A Bit More Precise

When computer resources and time are not an issue, you can choose to practically force RMA2 to meet your convergence criterion, or die trying. By choosing a large number for maximum iterations, such as 50, it is very likely that, if your convergence criterion is not unrealistically strict, RMA2 will converge to a solution before the maximum number of iterations are reached.

Convergence Problems

If the solution has not converged satisfactorily, but appears to be getting close, you should increase the maximum number of iterations and try again. If the solution is *diverging*, check the following situations:

- Erroneous boundary conditions, such as erroneous values in the input signal or an improper flow angle.
- Check the turbulent exchange coefficients and Peclet number (see “Modeling Turbulence” on page 37).
- Check for possible wetting and drying problems. When wetting and drying, additional factors are involved (see “Wetting And Drying” on page 78).
- Poor mesh design (ill-formed elements, boundary breaks, large slope gradients, etc.)



FastTABS Note: If you are using FastTABS version 3.0 or later, try the **Model Check** option in the File menu, and the **Mesh Quality** option available in Display Options, to find areas which may lead to problems during the RMA2 simulation.

Time Step Control

Timing is introduced into the simulation when the boundary conditions (head, velocity, discharge, wind) vary in time. This is known as a Dynamic, or Unsteady State simulation. The time step you use depends upon several factors, as described in “Selecting A Time Step Interval” below.

Selecting A Time Step Interval

Although RMA2 uses an implicit solution scheme, some experimentation is usually required when establishing the delta time step for dynamic simulations. In general, start with a value appropriate for your type of computations, and increase the delta time step to the largest value that is numerically stable and physically representative of the problem. For example, one hour time steps can be used in some tidal problems, even when amplitude is 6 to 8 feet, with a good reconstitution of observed gage records and current patterns.

Typically, a diurnal tide can use a 1 hour time step, while a semidiurnal tide or mixed tide requires a 0.5 hour time step.

The required computational time interval (time step size) may be dependent upon element sizes, strength of flows, flow patterns, and the rate of change in boundary conditions. Many tidal studies with this type of model employ 30 minute time steps satisfactorily, but experimentation should be used to ensure that the time step you select is appropriate for your problem. One approach to selecting this interval is to run a test case in which the time step size is reduced until the solution does not change, and use this interval for the time step.

For dynamic simulation runs, the computational time interval should be as large as possible to capture the extremes of the dynamic boundary conditions and maintain numerical stability. Traditionally, this time interval varies between 0.25 hours and 1.0 hour for tidal boundary conditions at the coasts of North America, provided that the hydrodynamics do not include complex features, such as wetting and drying, etc.

To reduce the computational time taken to complete a simulation, the time step interval should be as large as possible, while small enough to still accurately simulate the hydrodynamics of the modeled area. The interval should be small enough to

- Capture the extremes (highest and lowest peaks) of the boundary condition signal. If the interval is too large, the peaks of the signal may be missed.
- Gradually adjust to any wetting and drying conditions.
- Accommodate rapid changes in water surface elevation.

Setting the Computational Time Interval

There is only one time step used when running a steady state simulation. The computational time interval (DELTA) and the total run time (TMAX) should be set to zero on the TZ card.

Coding for dynamic (unsteady) flow is only a slight extension of coding for steady state flow. The key is in the timing information. You provide the computational time interval and total simulation time on the TZ card, and dynamic iteration control on the TI card. END cards are used to mark the end of boundary condition data for a dynamic time step.



Tip: The time step interval can be modified in mid-simulation using the REV card or BCC card. This ability is useful for refitting the time step to better approximate certain situations (as listed above) which may occur during the simulation.

Total Simulation Time

The total number of hours you want to simulate is the *simulation time*. You specify the time in hours on the TZ card.

Steady State Simulation Time

There is no time interval used when running a steady state simulation. The computational time interval (DELTA) and the total run time (TMAX) should be set to zero on the TZ card.

Dynamic Simulation Time

Dynamic simulations span a predetermined period of time. Specify the number of hours for the simulation on the TZ card.



Note: Remember that total simulation time is not just the time you consider for results, but should also account for the “spin-up” time

Simulating With Tides

For an estuarine model study, it is necessary to provide one or more boundary locations as a water surface elevation (such as a tidal boundary) which changes with time. For information on providing these boundary conditions, see “Providing Dynamic Boundary Conditions” on page 30.

If you do not have access to good prototype tidal elevations, then one alternative is to access or generate synthesized harmonic tidal data. There are several software packages available which will generate harmonic tidal data at most USGS station locations. The main harmonic constituent in a real world tidal cycle typically are the principal lunar semidiurnal tide (M_2) component, which has a period of 12.42 hours, and the O1, P1, and K1 diurnal components, which have periods of 25.82, 24.07, and 23.93 hours respectively.

For additional information on tides, see The Tidal Hydraulics Engineering Manual.

Specifying Boundary Conditions

Boundary conditions are required to drive RMA2 throughout a simulation. They are constraints which are applied along the flow boundaries of the solution domain, and required to eliminate the constants of integration that arise when we numerically integrate the governing equations to solve for u , v , and h in the interior of the solution domain.

External boundary nodes along the downstream end of the network are typically assigned a water-level (head) boundary condition. Also, boundary nodes along the upstream end of the network are typically assigned an exact flow or discharge boundary condition. Each side wall of the network is automatically assigned a Parallel Flow Boundary Condition (i.e., slip flow) which allows the program to calculate the velocity adjacent and parallel to the side wall as well as the flow depth there.

Boundary conditions may be specified on a per-node basis, along the edge of an element, or across a continuity check line. No special equations are required for boundary nodes. The use of a boundary condition specification removes either the depth, or one or both of the velocity components from the computations, and the program expects those values to be entered as boundary input data.

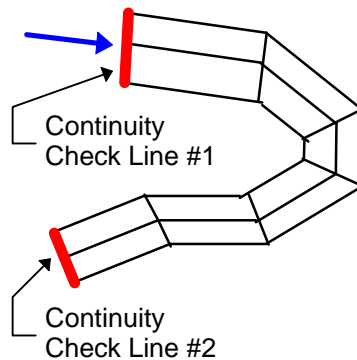


Note: All boundary conditions hold from one time step to the next unless they are specifically modified using a REV card. RMA2 does not permit a new boundary condition to be specified in mid-run, nor does it allow a change in the type of boundary condition.

Providing Dynamic Boundary Conditions

Dynamic simulations are used to model situations where water levels, flow rates, etc., can change over time, such as an estuary where ocean tides influence the water conditions. Therefore, dynamic simulations can have different boundary conditions specified at each time step.

The boundary conditions are provided in the run control file using the boundary related data cards. For each dynamic time step, a boundary condition should be provided for each inflow, outflow, or water surface condition that has changed from the value supplied in the previous time step. The example below illustrates the use of boundary conditions assigned by continuity check line.





Example:

```

.
.
CO    HOUR 1.5
BQL  1  20.0  6.11  Check line #1, angle at 350 degrees
BHL  2   3.0          Check line #2
END of hour 1.5
CO    HOUR 2.0
BQL  1  30.0  6.11  Check line #1, angle at 350 degrees
BHL  2   3.24       Check line #2
END of hour 2.0
.
.

```

To signal that all information for a dynamic time step has been entered, an END card is used to mark the end of information for the time step.

Types Of Boundary Conditions

There are several boundary conditions from which to choose to be specified at each node:

- Parallel Flow Boundary Condition (i.e., slip flow)
default for boundary nodes
- Flow Boundary Condition
- Water-Level Boundary Condition
- Stagnation Point
- Non-Reflecting Boundary Condition



Tip: The most popular set of boundary condition specifications is a downstream water surface elevation, and an upstream discharge (flow). However, the circumstances and field data availability may force you to alter this arrangement. For instance, a non-reflecting boundary condition may prove to be beneficial during a tidal simulation if the mesh domain does not extend to the head of tide.



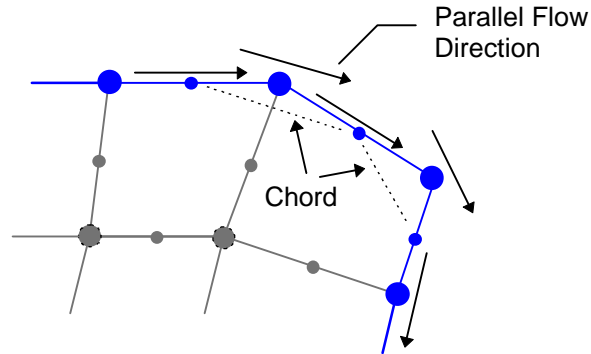
Note: The use of a boundary condition specification at a point removes either the depth, or one or both of the velocity components from the computations at that point, and the program expects those values to be provided as input data or otherwise prescribed. The type of information removed depends upon the type of boundary condition specification you choose.

Parallel Flow Boundary Condition

The flow at all boundary nodes defaults to parallel flow (or slip flow). Parallel flow occurs at the nodes which differentiate between the wet and dry areas of the mesh. RMA2 is free to calculate the velocity adjacent and parallel to the side wall, as well as the depth at the side wall. This means that the depth can become infinitely high at the side walls if the conditions of the simulation dictate.

Parallel Flow At Mesh Exterior Edges

At external corner nodes, parallel flow is the flow of fluid parallel to the *chord* connecting the midside node of each of the two adjacent elements. For midside nodes of straight sided elements, parallel flow is simply parallel to the element edge.



If an external two dimensional boundary has no explicit specification, and is made up of a series of straight element sides with arbitrary sharp breaks at the corner nodes, then the velocity vectors at the corner nodes are adjusted such that local water fluxes are balanced around the corner nodes. When the depth along the boundary is constant the direction of flow is parallel to the chord between the adjacent boundary midside nodes.

The parallel flow angle may be altered by curving the element edges, thereby allowing more realistic vector angles without the computational expense of adding resolution.



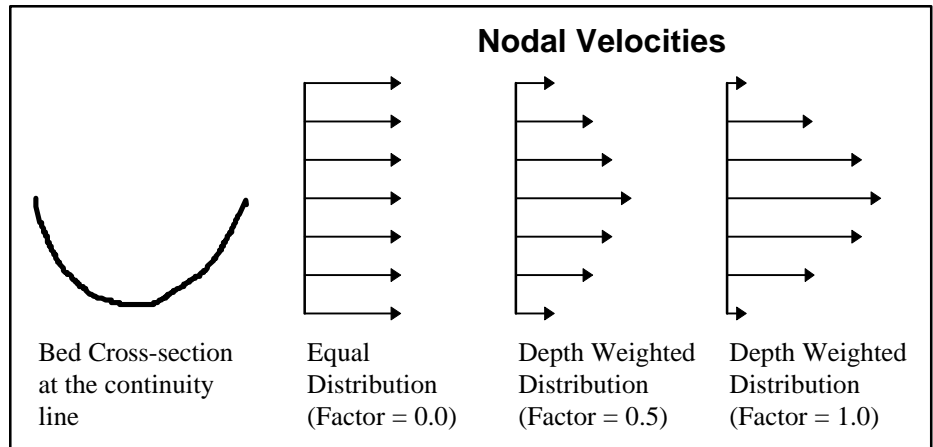
Note: The default parallel flow angle can be overruled at a node by using a BA card with the *by node* option, N.

Flow Boundary Condition

A flow boundary condition is specified as either Total Flow or Velocity. They are normally provided at inflow locations, and are used to specify the amount and direction of the fluid entering the mesh.

Total Flow

The most popular and straightforward way to specify a flow boundary condition is to provide the total flow crossing a particular continuity line with the BQ card using the *by line* option, L. The total flow (cubic units per second), flow angle (radians), and *distribution factor* at the line are specified on this card. The *distribution factor*, which can be a real number between 0.0 and 1.0, is used to weight the nodal *x* and *y* velocity components according to the nodal depths across the continuity line.



A factor of 0.0 is the default. A factor of 1.0 will weight the velocities according to nodal depth across the continuity line..



Note: The *total* flow crossing the continuity line is the same regardless of the value of the distribution factor.

Another way to get water in and out of elements is by using the BQ card with the *by element* option, E. This method is useful when attempting to include rainfall and evaporation in the simulation. See “Adding Rainfall And Evaporation” on page 84 for more information.

Velocity

The x and y components of velocity may be provided for each node across the boundary with the BCN card. The flow angle is determined by the resultant of the x and y components. Since the values of the x and y components of velocity and the angle are supplied, and the model solves for depth, the total flow across the boundary will fluctuate as dynamic water surface boundary conditions take effect.

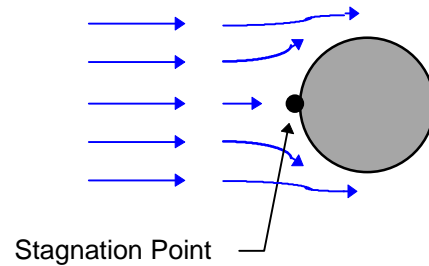
Water-Level Boundary Condition (Head)

A water surface elevation boundary condition, or *head*, is normally provided at the tailwater location. The most popular technique for assigning the head boundary condition is to specify the head across a *Continuity Check Line* with a BH card using the *by line* option, L. An alternative method is to supply the head at each *boundary node* with the BH card using the *by node* option, N, or the BCN card. If the head is specified in one of these ways, the model will solve for the x and y components of velocity, and the flow angle at the associated node(s).

At an exit boundary, RMA2 adjusts the water depth as needed to obtain the optimum solution to the finite element equations. The minimization principles in the solution technique reach optimums when all heads, even those along the exit boundary, can be adjusted individually to preserve overall network coherence. The result is water surfaces at the exit boundary nodes that may differ slightly from the coded boundary conditions.

Stagnation Point Boundary Condition

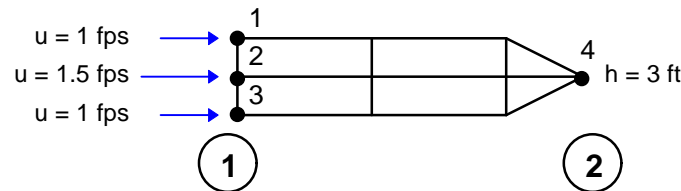
Occasionally, it may be appropriate to use the stagnation boundary condition, which means that both the x and y velocity components are set to zero at the specified node. One situation where this may be useful is when there is a perpendicular flow at a boundary, such as at a bridge pier. The perpendicular flow at such a point is zero. The zero velocity components are explicitly assigned for that node on the BCN card.



Typical Boundary Condition Examples

Nodal Assignments

1. Specifying x and y velocity at a node (BCN card)
2. Specifying water surface elevation (head) at a node (BCN card)



Example:

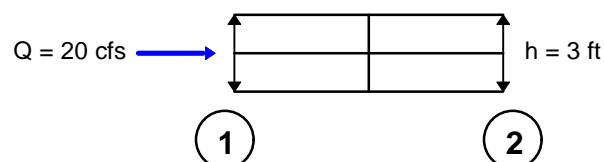
```

.
.
BCN 1  11000  1.0  0.0  0.0    at Node 1
BCN 2  11000  1.5  0.0  0.0    at Node 2
BCN 3  11000  1.0  0.0  0.0    at Node 3
BCN 4   200   0.0  0.0  3.0    at Node 4
END of time step
.
.

```

Assignments Along Boundary Lines

1. Specifying discharge along a boundary line (BQ card with the *by line* option, L)
2. Specifying water surface elevation (head) along a boundary line (BH card with the *by line* option, L)





Example:

```

.
.
BQL  line1  20.0  0.0
BHL  line2   3.0
END of time step
.
.

```



Note: Alternatives to the BCN card are sets of cards (BA - BS or BQ - BH) on which the same three boundary parameters are coded as on the BCN cards, but they are coded in alternate formats which may be more convenient. The first set allows an azimuth to be used along with a current or discharge to establish the inflowing velocity components: BA card = azimuth of the boundary velocity vector, BS card = speed of the boundary velocity, and BH card = the water-surface elevations (The BQ card can be used in place of the BS card). The value of NFIX is determined by the program based on card types present. The BA card should precede the others and azimuths on it will be used to calculate either velocity components or unit discharge components until another BA card is read. Only those values which differ from previous values must be changed.

Boundary Condition Gotcha's

Overspecification

An error will occur if you attempt to assign more than one type of boundary condition to the same location.

It is possible to *overspecify* the boundaries of the problem. For example, do not code water elevation and flow velocity at the same node because that attempts to force both a slope and the water discharge. Theory does not allow such in subcritical flow, nor can it be accomplished in a flume unless the roughness is a dependent variable. Also, overspecification of the problem is more likely when including internal nodes in the boundary conditions data set. Therefore, establish model performance criteria very carefully to make certain the results are not contaminated when boundary specifications are being assigned to internal nodes.

Too Close For Comfort

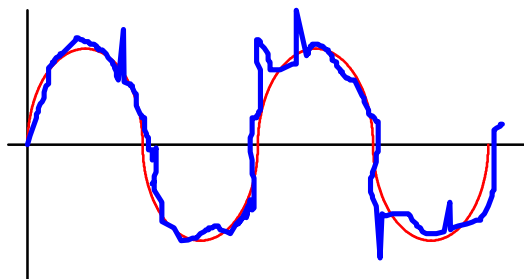
It is advisable to place all boundary conditions as far away from the area of interest of your study as possible and reasonable. If boundary conditions are close to your study area, the results which interest you can be unduly influenced by the values assigned for those boundary conditions.



Tip: One Dimensional Elements are useful when you want to remotely place boundary conditions without adding a lot of resolution.

Noisy Boundary Condition Signal

A noisy/sporadic boundary condition signal with spikes is common with collected prototype data. Unless the signal is filtered or hand smoothed, RMA2 may diverge.



Dry Boundary Condition

If any node associated with a boundary specification becomes dry, the simulation will very likely die. Unfortunately, there are few if any diagnostics reported for this error.

Too Quick, Too Fast

If the change in elevation or discharge at the boundary varies too quickly between successive time steps, RMA2 may diverge. Generally, the change in water surface elevation should not exceed 1 foot per 1 hour time step, and the change in discharge should not exceed approximately 5,000 cfs per 1 hour time step.

See “Time Step Control” on page 28 for more information on time steps.

Bed Friction And Resistance To Flow

The bed friction energy transfer computation, or *roughness*, is one of the primary verification tools for RMA2. Changing the bed friction provides some control over the fluid velocity magnitude and direction.

The bottom shear stress, T , is defined as

$$T = \rho g R S$$

where

ρ is the density of water, g is the acceleration of gravity, R is the hydraulic mean radius, and S is bed slope.

Bed friction is typically calculated with Manning’s equation if the input roughness value is < 3.0 , otherwise a Chezy equation is used. By far, the popular choice is Manning’s n -value, and these roughness values may be assigned globally throughout the mesh by material type, or on the elemental level.

The Manning’s equation for uniform flow is

$$V = 1.49 * \frac{R^{2/3} * S^{1/2}}{n}$$

where V is the velocity, and n is the n -value

By solving the Manning’s equation for S and substituting the results into the equation for bottom shear stress, we have

$$T = rg \left(\frac{n}{1.49} \right)^2 \frac{V^2}{R^{1/3}}$$

Since the hydraulic mean radius, R, is the cross section area divided by the wetted perimeter of the cross section is approximately equal to the depth for wide channels, the final form of the components for the bottom shear stress in terms of Manning's equation is

$$T_x = rg \left(\frac{n}{1.49} \right)^2 \frac{u \sqrt{u^2 + v^2}}{h^{1/3}}$$

$$T_y = rg \left(\frac{n}{1.49} \right)^2 \frac{v \sqrt{u^2 + v^2}}{h^{1/3}}$$

where h is the channel depth.



Note: RMA2 provides the means to input only the bottom roughness, not the side wall roughness.



Tip: Because there is no wall roughness in RMA2, you may need to exaggerate the bed roughness on the elements forming the edge of the waterway in order to approximate the wall roughness.

Defining Roughness

There are two ways to assign roughness:

- Manual Roughness Assignment
- Automatic Roughness Assignment

Manual Roughness Assignment

You may assign particular n-values using the HN card or EV card. Assignments can be made by individual element or by material type. Global assignment is also possible.

Automatic Roughness Assignment

You may choose to have RMA2 automatically assign roughness values for each iteration according to the calculated depth at each Gauss point with the RD card (see “Automatic Friction Assignment” on page 77).

Modeling Turbulence

All forms of numerical models require some form of stabilization. The Galerkin method of weighted residuals employed by RMA2 does not include any inherent form of stabilization other than the eddy viscosity terms. The Galerkin formulation

requires a certain amount of turbulence to achieve stability and not impair the solution. However, one may get useful solutions by specifying a higher turbulence value than is physically justified. This technique will produce a stable model without the need to add too much resolution.

What Is Turbulence?

The fluid momentum transfer due to exchanges of fluid masses moving at different speeds is called turbulence exchange. It is approximated in the governing equations by multiplying a turbulent exchange coefficient, E , by the second derivative of the velocity with respect to the x and y directions. The English units for turbulence are lb-sec/ft^2 . The program allows for the exchange coefficient to be specified in a local coordinate system for each element

Although it is difficult to establish the value for E , analogy with physical conditions suggests that turbulence exchanges depend on the momentum of the fluid and the distance over which that momentum is applied, divided by the fluid velocity and the surface area of the element. Therefore, as the element size increases, E should increase, or as the velocity increases, E should increase, in order for the rate of turbulent exchange to remain constant.

Specifying Turbulence

Turbulent exchanges are sensitive to changes in the direction of the velocity vector. Conversely, small values of the turbulent exchange coefficients allow the velocity vectors too much freedom to change directions in the iterative solution. The result is a numerically unstable problem for which the program will diverge rather than converge at a solution. The recourse is to continue increasing the, E , until a stable solution is achieved.

There are two basic ways to control the turbulent exchange coefficient, E .

- Assignment By Material Type
- Automatic Assignment By Peclet Number

Assignment By Material Type

The first way to assign the turbulent exchange coefficient, E , is to assign a particular value for each individual material type with the EV card.

As a guideline for selecting reasonable values for the turbulent exchange coefficients for a given material type you should:

- determine a representative length of the elements within the material type
- estimate a maximum streamwise velocity for the given material type
- solve the Peclet equation for E , given a Peclet number of 20

Several modelers choose to automate this process. A utility program called **MAKE_EV** will calculate the average elemental size of each material type and create a table of recommended eddy viscosity values for each material type in the mesh.

The table below provides some representative ranges of turbulent exchange coefficients.

Type of Problem	E, lb-sec/ft ²
Homogenous horizontal flow around an island	10-100
Homogenous horizontal flow at a confluence	25-100
Steady-State flow for thermal discharge to a slow moving river	20-1000
Tidal flow in a marshy estuary	50-200
Slow flow through a shallow pond	0.2-1.0

Automatic Assignment By Peclet Number

The second way to assign the turbulent exchange coefficient, E, is to allow the model to automatically adjust E after each iteration, based upon a provided Peclet number, which is based upon the unique size and calculated velocity within each element. The Peclet number defines the relationship between the velocity, elemental length, fluid density, and E.

Recall the formula for Peclet number (*P*), where *P* is recommended to be between 15 and 40.

$$P = \frac{\rho u dx}{E}$$

where:

Coefficient	English Units	Metric Units
ρ = fluid density	1.94 slugs/ft ³	1 grams/cm ³
u = average elemental velocity	fps	mps
dx = length of element in streamwise direction	ft	m
E = eddy viscosity	lb-sec/ft ²	Pascal-sec

For details on how to assign automatic Peclet values, see “Automatic Turbulence Assignment” on page 78.

Results Listing Control

Results listing files contain data from the RMA2 simulation run. There are two primary types of results listings:

1. Full Results Listing
2. Summary Results Listing

Full Results Listing

To create a full results listing file, set the variable IOUT on the \$L card to any positive number. Results are written to the full listing depending upon settings on the TR card (see “Customizing The Full Results Listing File” on page 68 for more information).

Coldstart

For a Coldstart, it is recommended that new users request a complete full listing at the end of each time step.



Example: To obtain a full results listing when Coldstarting, use the TR card set as follows:

```
.
CO   Coldstart listing, write results every time step
TR   1  -1  1  1  0
.
```

A complete listing will be useful when diagnosing problems with a simulation.

Hotstart

For a Hotstart, it is recommended that new users request a full listing at the end of each time step. A complete listing of Hotstart initial conditions is virtually the only way to verify that the model restarted correctly.



Example: To obtain a full results listing of initial conditions when Hotstarting, use the TR card set as follows:

```
.
CO   Hotstart listing, write results every step
TR   2  -1  1  1  0
.
```



See Also: “Results Listing Files” on page 20, “Understanding The Full Listing File” on page 49, and “Customizing The Full Results Listing File” on page 68.

Summary Results Listing

To create a summary results listing file, set the variable ISPRT on the \$L card to any positive number, and supply a list of nodes with TRN cards. Summary results are written at the completion of each time step.



Tip: The summary listing for specified nodes provides a means to use RMA2 results in spreadsheet software for graphing and further analysis.



See Also: “Results Listing Files” on page 20.

Stopping The Simulation

Aside from turning off your computer, there are two ways to stop a simulation run:

1. Normal Run Completion
2. Immediately Stopping A Simulation by interrupting from the keyboard

Normal Run Completion

The intended way for a simulation to end is for it to run to completion. If the simulation stops normally, any one of the following must have been satisfied:

- Total Simulation Time
- Total Number Of Time Steps
- A STOP Card was read
- A Diverged Simulation

Upon normal completion, RMA2 will close all input and output files, and erase any temporary files it may have written during the run.

Total Simulation Time

RMA2 will stop after completing TMAX hours of simulation. TMAX is specified on the TZ card.

Total Number Of Time Steps

RMA2 will stop after completing NCYC time steps. NCYC is provided on the TZ card.

A STOP Card

A STO card (Stop card) should always be included in the run control file. The STOP card tells RMA2 that there are no more data to read and the simulation will terminate.



Note: The STOP card should always be preceded by an END card.



Tip: Normally, a STOP card is placed at the end of the run control file. However, when debugging, it may be useful to place a STOP card at an earlier point in the run control sequence, say, after the first time step for example to compute steady state results only.

A Diverged Simulation

RMA2 will stop whenever the convergence parameters indicate a maximum change in depth in excess of 25 feet from one iteration to the next.

Immediately Stopping A Simulation

If you need to stop the simulation before it has completed, you can interrupt RMA2 using the means available on your system to kill a process, typically by pressing Control-C, or Control-Break, at the keyboard.



Note: If the simulation runs to normal completion, all associated scratch files will be eliminated and all input/output files will be closed normally and saved if applicable. However, If the simulation is abruptly terminated (Control-C from the keyboard, or the CPU limit is exceeded, etc.), large scratch files may be left resident on your disk. The scratch files are typically assigned to unit 9 in batch mode, or, in

interactive mode, the filenames may start with a capital 'H'. If these file are left resident on the disk, you should delete them.

Any Comments?

RMA2 allows the use of comments in the run control file. Comments may be placed anywhere in the file, except on the first line, where a Title card is required (see “Using Titles” on page 22).

To include comments, use the CO card.

Verifying The Simulation

Verification: A Process

Some numerical modelers inaccurately refer to calibration and verification as a two-step process. Using this terminology, adjustments are made to model coefficients and inputs so as to optimize agreement between model and observed prototype data during the calibration step. Then the model is run to attempt reproduction of a different set of prototype data without further model adjustment. If the second run is satisfactory then the model is considered verified.

This procedure sounds imminently reasonable, but experience suggests that it is a naive approach and is definitely bad modeling practice. The reasons are these:

1. All field data are in error, and sometimes dramatically in error. Thus, they are not an absolute standard (a necessary characteristic of calibration), and may not be even a good approximate standard.
2. Field measurements include a variety of effects that may not be reproduced in the model (for example, groundwater flow into the model)
3. Conditions often change between field surveys, mandating that coefficients should also change (for example, differences in bed forms at different flows may dictate a change in bed roughness coefficients)
4. Most natural waterways cannot be adequately characterized by two field data sets. Five or ten may be needed, but available resources usually limit the field data to a non-ideal level.
5. If the model reproduces one field data set adequately, but not the second, you must decide whether to:
 - a) Proceed with modeling, conceding an incomplete verification.
 - b) Continue adjusting/revising to obtain a balanced quality of reproduction.
 - c) Conduct a re-analysis/re-collection of field data.

Verification is a multi-step process of model adjustments and comparisons, leavened with careful consideration of both the model and the data. It is not a simple two-step procedure, and the term calibration should not be used. The purpose of numerical modeling, as stated by W. A. Thomas of WES, is to “gain insight, not answers”.

Checking For Continuity

Continuity refers to conserving mass within the model. The objective when simulating is to retain the correct amount of fluid flow from one point to the next, within a tolerance of about plus or minus 3%. *Continuity check lines* provide the means to determine if your model is locally maintaining mass conservation at a given location.

Continuity check lines are typically used to estimate the flow rates and serve as an error indicator. The RMA2 model globally maintains mass conservation in a weighted residual manner. Locally, continuity check lines can be used to check for apparent mass changes in a different way, by direct integration. Large discrepancies between the results of these two methods indicate probable oscillations and a need to improve model resolution and/or to correct large boundary break angles.

Although continuity checks are optional, they are a valuable tool for diagnosing a converged steady state solution. For steady state, the continuity check lines should represent *total flow in equals total flow out*. However, if the continuity checks indicate a mass conservation discrepancy of $\pm 3\%$, you may want to address the resolution in the geometry. Large mass conservation discrepancies can lead to difficulty when the hydrodynamics are used for transport models, RMA4 and/or SED2D.

Of course, all this assumes a converged solution.

Continuity Check Lines

Continuity check lines provide two services:

1. Provides a convenient means to assign a boundary condition.
2. Allows for reporting the total flow passing through the line.

Continuity check lines are typically assigned using a GC card, by selecting corner nodes from the right bankline to left bankline, facing downstream. They should be oriented perpendicular to the direction of flow. Avoid placing check lines in an area where an eddy may form.

After The Simulation

Critical Check Points

When you run a simulation, you should first determine if the solution is numerically stable and realistic. Critical check points include the convergence parameters, and total flow across continuity check lines found in the full results listing, and examination of the solution by means of a velocity vector plot and water surface contour plot.

Continuity Checks And Conservation Of Mass

Poor continuity, or “leaking”, may be the result of poor mesh resolution, sharp boundary breaks, or severe curved edge related problems. The difference in convergence parameters should decrease with each iteration within a given time

step (acceptable exceptions would involve wetting and drying or revisions of the network within a time step).

Continuity checks can be used in steady state to determine if boundary conditions are holding properly and whether mass is conserved. Graphical analysis of a stable model will not have abnormal vector behavior or spurious contours.



Tip: In general, curved short element boundary edges improve continuity.

Effects Of The Initial Conditions

For dynamic runs, the initial conditions can adversely affect the results during the first part of a simulation. This is because the model is “shocked” by the initial conditions if they are not very close to what is expected.

Check to insure that the time interval you are concerned with is not affected by the initial conditions. A time series plot of both water surface and velocity at a valid control point will illustrate when the model begins to repeat the solution. This technique is valid only if you are running a repetitive boundary condition.



Tip: For a tidal simulation, use a smooth sinusoidal repetitive water surface elevation boundary condition signal as input to estimate spin-up time.

Is The Model Realistically Simulating Wetting And Drying?

A network designed such that the edges of elements fall on contours which are critical to wetting and drying will usually perform satisfactory. However, a poorly designed network constructed without attention to wet/dry boundaries will oscillate and sporadically pull elements in and out of the solution.

A quick check point is to skim the total number of active nodes for each iteration. A more specific indicator is to look in the full results listing to see exactly which element numbers have been added or eliminated per time step. In addition, graphical analysis will display the network as it exists for that time step. Instability may result when the dried edge results in a jagged boundary.

For wetland applications it is advisable to run the marsh porosity technique in conjunction with traditional wetting and drying by element. The marsh porosity technique allows elements to transition gradually between wet and dry states.

If Your Verification Fails

If you are experiencing difficulties verifying the model, you should check the items listed below.

Wrong Choice Of Model

If you attempt to model three dimensional current phenomenon with a two dimensional depth averaged model, the situation is basically hopeless. You should use a three dimensional model such as RMA10.

Geometry Problems

Geometry is more than just the assemblage of nodes and elements, it incorporates the idea of study design and model domain. There are several geometry issues which may cause verification problems.

Element Problems

Check for *ill-formed elements* as described in the GFGEN Geometry manual.

Boundary Location Problems

Boundary conditions should be located away from the study area where data is reliable. Look for locations such as a dam, lock, head of tide, or null point in the flow. Do not assign them at locations with high flow rates.

Wetting And Drying Problems

Incorrectly simulating wet/dry conditions can lead to numerical shocks, large boundary break angles, and/or water storage problems. See “Wetting And Drying” on page 78 for details on wetting and drying.

Water Storage Problems

Too little, or sometimes too much, tidal storage, or backwater storage, is present in the geometry. Storage areas can affect velocities. Add, or remove, storage areas as necessary to match the real world system.

Bathymetry Problems

Be sure the mesh is in agreement with the prototype conditions which are being simulated. Also, be sure the bathymetry is fairly smooth (maybe you can swap an element edge between corner nodes). RMA2 is based upon the Mild Slope Assumption (see “Limitations” on page 2).

Resolution Problems

Resolution must be appropriate to the phenomena modeled. To model an eddy, for example, you must have enough resolution to capture the eddy.

Boundary Condition Problems

Use good, reliable boundary data at all boundaries.

The boundary conditions must match the conditions during which the field data was collected.

Choose an appropriate type of boundary condition for the boundary location (elevation, discharge, rating curve, wind speed and direction, fluid speed and direction, tidal reflection/absorption)

Check for erroneous boundary conditions, such as a sporadic value in the input signal or an improper flow angle.

Roughness Problems

Roughness affects the water surface profile (steady state river simulation). It can affect the speed of the flow in the river passage. Wetting and drying may be affected.

Because there is no wall roughness in RMA2, you may need to exaggerate the bed roughness on the elements forming the edge of the waterway in order to simulate the wall roughness.

Eddy Viscosity Problems

Turbulence issues can create problems during a simulation. Since viscosity is involved in the equations of motion, it effects the velocity distribution, which will effect the depth, which will determine the effects of roughness, which may in turn effect the phase and amplitude of the tidal signal, etc.

If the eddy viscosity is too high, velocity magnitudes will appear uniform across the channel. Eddies will not form in the waterway if the eddy viscosities are too high. Picture a high viscosity value having the effect of turning the water to “syrup”.

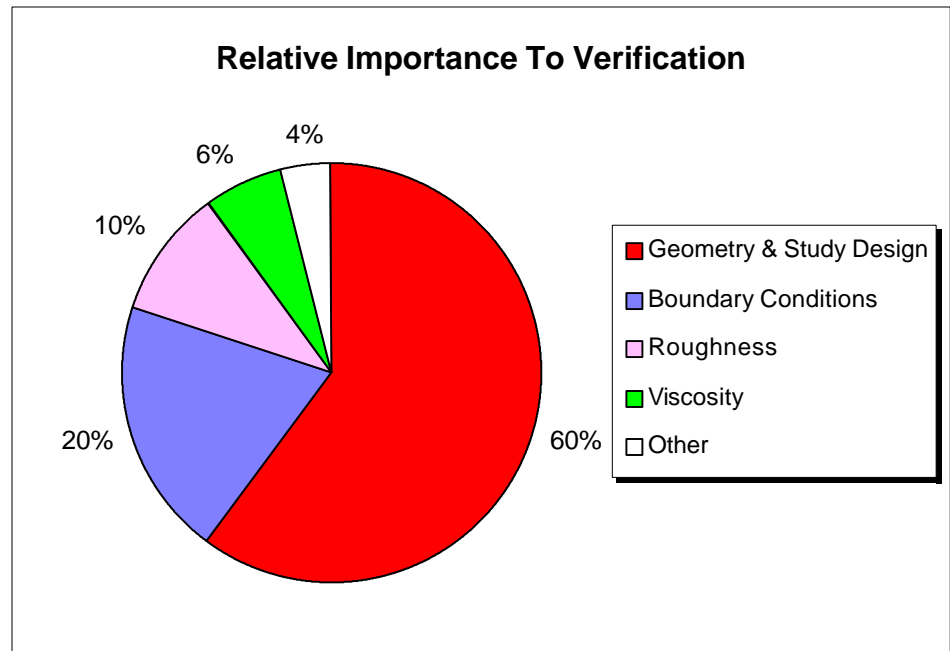
If the eddy viscosity is too low, the velocity contour patterns will appear erratic. The textbook case of a two delta-X oscillation, defined as an unrealistic neighboring high/low/high/low contour pattern, may arise.



Note: Eddy viscosity and Peclet number are inversely related. See "Automatic Assignment By Peclet Number" on page 39.

Influence On Verification

This chart illustrates the approximate relative importance to the simulation of the different aspects of an RMA2 simulation study, as described above. As you can see, the structure of the geometry and overall study design are the most significant, followed by the boundary condition assignments.



Interpretation Of Results

Overview

RMA2 is capable of generating more data in one dynamic simulation than a single person could absorb in a month. The key to understanding the results from the model lies in requesting to see key segments of information, such as: initial conditions, convergence parameters, steady state continuity checks, and mini-statistics. Most of these key segments are located in both the full results listing file and the screen output. Additional global and detailed information can be obtained from post-processing utilities, and/or a graphical user interface such as FastTABS.

To become an expert RMA2 user, you should be aware of what is included within the three types of RMA2 output.

- Understanding The Full Listing File
- Understanding The Summary Listing File
- Interpreting The Solution

Understanding The Full Listing File

The Full Results Listing file is the most comprehensive analysis tool available in RMA2. It may seem overwhelming with all its lists of numbers and computer lingo, but it's really not.

The full results listing file can be separated into these sections:

- Model Information
- Input Interpretation
- Marsh Porosity Parameters
- The Element Connection Table
- Echo of Boundary Conditions
- Nodal Specifications, or, Initial Conditions (Hotstart)
- Steady State And Dynamic Simulation Progress And Statistics
- Steady State And Dynamic Simulation Nodal Results

The easiest way to locate these sections is to place the listing file into a text editor which has a search function and search for keywords which are unique to each section. These keywords are presented below within the description of each section.

Model Information

General model information appears first in the full results listing. This information is always included and cannot be switched off.

File Names

The file names supplied to an interactive RMA2 simulation at startup are listed at the beginning of the full results listing file. The types of files, of course, depend upon the run options which were selected in the run control file.



To find this section, look at the beginning of the full results listing file.

Version Number And Modification Date

The version number and last modification date are provided here. This information can be very important when obtaining technical support.



To find this section in the full results listing file, search for the text “**RMA2 VERSION**”, with 3 spaces separating.

Here is a typical representation:

RMA2 VERSION 4.30 1-D AND 2-D CAPABILITY. LAST MOD DATE 08-25-1995

Program Dimensions

The program array dimensions used for the simulation are listed here. If RMA2 cannot run using your data, you should verify that the program dimensions are large enough to accept your data. Generally, check the number of **nodes** and **elements**, and also the **front width**.



To find this section in the full results listing file, search for the text “**DIMENSIONED**”.

Here is a typical representation:

THIS PROGRAM IS DIMENSIONED AS FOLLOWS	
MAX NO. OF NODES	20000
MAX NO. OF ELEMENTS	6500
MAX NO. OF EQUATIONS	40000
MAX FRONT WIDTH	600
MAX NO. OF CONTINUITY CHECK LINES	100
MAX BUFFER SIZE	100000
MAX PRINT-SUMMARY BUFFER	50000



Tip: The dimension labeled **MAX BUFFER SIZE** influences the number of temporary files which are written at each iteration. You may be able to improve the performance of RMA2 by increasing this dimension in the RMA2 source code. For details, see “Reducing The Number Of Temporary Files” on page 170.

Input Interpretation

An interpretation of the data cards used in the run control file appears in the full results listing after the Model Information section. This information is always included and cannot be switched off.

This section includes

- Run Control Parameters
- Geometry Input Summary

Run Control Parameters

This section of the full results listing file contains several areas pertaining to the different aspects of run control. It includes information such as:

- Selected options
- Logical unit number assignments for files
- The machine identifier
- Various geometry parameters
- Timing controls
- Peclet number and eddy viscosity information
- A list of assigned continuity check lines.



To find this section in the full results listing file, search for the text “**RUN CONTROL PARAMETERS**”.



Note: This section may appear in multiple locations if revisions (REV card) have been specified.

Geometry Input Summary

This small section of the full results listing file is to inform you as to what RMA2 has read from the GFGEN binary geometry file. The geometry title and the GFGEN version number are provided here, but most important are the network statistics which you use to verify that RMA2 has properly read your geometry data.



To find this section in the full results listing file, search for the text “**NETWORK INPUT**”.

Here is a typical representation:

```
.....NETWORK INPUT COMPLETE.....
```

```
MAX ELEMENT NUM = 200
MIN ELEMENT NUM = 1
MAX MATERIAL NUM = 3
MIN MATERIAL NUM = 1
MAX NODE NUM = 640
MIN NODE NUM = 1
```

Marsh Porosity Parameters

An echo of the nodal Marsh Porosity parameters: AC1, AC2, AC3, and AC4. This information will always be included in the full results listing file if the marsh porosity option is activated with the DM card. If RMA2 was run without a DM card, this section will contain only the text **MARSH ELEMENTS INOPERATIVE**.



To find this section in the full results listing file, search for the text “**MARSH ELEMENTS**”.

Element Connection Table

The element connection table contains detailed information on all the elements in the mesh. This information is omitted from the full results listing file if IPRT on the TR card is set to zero.

The element connection table lists all of the element numbers along with their associated node numbers, as well as the Coriolis (Latitude) values for each element. Additionally, this table will allow you to determine the nodes which make up an element, the material type, the elemental area, and any modifications which may have been made to the angle specified for the eddy viscosity tensor.



To find this section in the full results listing file, search for the text “**NODAL CONNECTIONS**”.



Note: An element with *negative* area indicates that its nodes are listed in reverse order. Be sure the nodes comprising the element are listed in a counter-clockwise order. An element with *zero* area may indicate that no eddy viscosity has been specified for the material type associated with the element. Alternatively, elements with zero area may be a control structure, junction, or an element whose material type is zero.



FastTABS Note: Safeguards within the FastTABS mesh generator help prevent you from making common network errors, such as, an element with a negative area (referenced in the “Full Results Listing” on page 39), and other element connection errors. However, these are indeed fatal errors which must be corrected.

Echo Of Boundary Conditions

This section of the full results listing file lists the boundary conditions that were specified for the simulation run. Boundary conditions defined as element flow, flow across a check line, and elevation along a check line can be listed here. The type of boundary condition is stated, then the associated parameters defining the boundary condition.



Note: If you are simulating using rainfall or evaporation, the flow rate value listed for boundary conditions defined as element flow for the affected elements will

be different from what you specified. The values have been converted to be in the units used by RMA2 during calculations.

This section may also contain information on any flow control structures that are defined.



To find this section in the full results listing file, search for the text “**CONDITION DEFINED**”.



Note: This section may appear in multiple locations if revisions (REV card) have been specified.

Nodal Specifications Or Initial Conditions

At this point in the full results listing file you will find either the Nodal Specifications or the Initial Conditions. What is located here depends upon the value of IPRT on the TR card.

Nodal Specifications

In the full results listing file, the nodal specifications include the x and y location of each node, its bottom elevation, and its network slope. Generally, a network slope value of zero indicates an interior node.

Additionally, the nodal specifications are useful for determining whether a particular node has a boundary condition assigned, and if so the type of specification. The type is listed as a number in the column marked **B-C FIX** (see the BCN card for a description of these numbers). If the type is zero, no boundary condition is assigned at that node. If a node has a boundary condition specified as flow, the flow rate in the x and y directions will be listed for the node. If the boundary condition is specified as an elevation, the elevation will be listed.

The nodal specifications also include one dimensional nodal cross-sectional information. The node surface width at zero depth, left and right side slopes, and storage width are included. For the two dimensional nodes, these data fields are zero.



To find this section in the full results listing file, search for the text “**NODAL SPECIFICATIONS**”.



Note: The Nodal Specifications only appear when IPRT on the TR card is set to 1.

Initial Conditions (Hotstart)

To assure that RMA2 Hotstarted at the correct location from the previous run, look in the full results listing file and verify that the **ACTIVE NODAL STATISTICS** for the first iteration of the Hotstarted run match the **ACTIVE NODAL STATISTICS** from the last iteration of the previous run.



To find this section in the full results listing file, search for the text “**INITIAL CONDITIONS**”.



Note: The Hotstart Initial Conditions only appear when IPRT on the TR card is set to 2. This section is only useful if Hotstarting the model.

Steady State And Dynamic Simulation Progress And Statistics

A steady state simulation is always performed when running RMA2. If a dynamic simulation is requested, the steady state results are used as initial conditions for the dynamic portion of the simulation.

This section contains information which shows how the model is being solved during the simulation. The information will appear both in the steady state and the dynamic sections of the Full Results Listing file. This section has the same format for both the steady state and dynamic portions of the simulation, so this text is satisfactory for describing it for both portions.

This information is always included and cannot be switched off. Detailed information about each iteration is provided. This is valuable information you can use to help find the cause of problems.

For each iteration, various information is written to the full results listing file. Each iteration will include the simulated hour and the iteration cycle. For steady state, the simulated hour is always zero.

Number Of Equations

The number of equations necessary to obtain a solution is reported here in the full results listing file. This number may change when wetting and drying are occurring. Remember that the maximum number of equations allowed is stated as **MAX NO. OF EQUATIONS** as described in “Program Dimensions” on page 50.



To find this section in the full results listing file, search for the text “**SYSTEM EQUATIONS**”.

Buffer Blocks

Before RMA2 can begin to solve the equations, it must build the solution matrix. If the value of **MAX BUFFER SIZE**, as described in “Program Dimensions” on page 50, is not large enough to allow RMA2 to store the matrix in memory, RMA2 must store the remainder of the matrix on disk as temporary files, or *buffer blocks*. The number of buffer blocks written is indicated at the beginning of each iteration.



To find this section in the full results listing file, search for the text “**BUFFER BLOCKS**”.



Tip: The dimension labeled **MAX BUFFER SIZE** influences the number of temporary files which are written at each iteration. You may be able to improve the performance of RMA2 by increasing this dimension in the RMA2 source code. For details, see “Reducing The Number Of Temporary Files” on page 170.

Front Width

The **MAXIMUM FRONT WIDTH** is listed immediately under the **BUFFER BLOCKS WRITTEN** section in the full results listing. The front width is indicated at the beginning of each iteration. Larger values of front width will increase computation time and may be indicative of a failure to properly reorder the mesh.



To find this section in the full results listing file, search for the text “**MAXIMUM FRONT WIDTH**”.

Here is a typical representation:

```
TOTAL NUMBER OF ACTIVE SYSTEM EQUATIONS = 1296
18 BUFFER BLOCKS WRITTEN, FINAL LQ SIZE = 434
82 IS MAXIMUM FRONT WIDTH
```

Convergence Parameters And Nodal Statistics

The convergence parameters and nodal statistics are written to the full results listing file, and to the terminal display, after every iteration. They inform you as to how well the model is performing at any given iteration. The convergence parameters and nodal statistics are normally examined together. In situations where the model is diverging, these sections can provide you with the location of the problem.

Here is a typical representation. This solution is diverging and the problem seems to be at node number 1.

```
RESULTS AT THE END OF      0 TIME STEPS...
TOTAL TIME =      .000000 HOURS... ITERATION CYCLE =    14

CONVERGENCE PARAMETERS

      DF      AVG  CHG      MAX CHG      LOCATION
      1      .0316      -22.9952      1      X-VEL
      2      .0191      1.9483      32      Y-VEL
      3      .0196      -.2465     7259     DEPTH

ACTIVE NODAL STATISTICS FOR THIS ITERATION

      NODE  XVEL-MAX      NODE  XVEL-MIN      NODE  YVEL-MAX      NODE  YVEL-MIN
      119      1.126      1      -21.617      1      12.016      119      -.930

      NODE  ELEV-MAX      NODE  ELEV-MIN      AVE-ELEV      NODES  ACTIVE
      31      835.875      6992      835.460      835.566      8751
```

The convergence parameters and nodal statistics can help you find problem areas in your mesh. If the solution is diverging, you can look at the convergence parameters at previous iterations to find the time when, and the nodal location where, the divergence first began to occur.

Convergence Parameters

The column marked **AVG CHG** is the average of all active nodes for each of the three degrees of freedom: *x* component of velocity, *y* component of velocity, and water depth. The column marked **MAX CHG** is the value of the maximum change between this and the previous iteration for each of the three degrees of freedom. The column marked **LOCATION** is the node at which the maximum change occurs for the three degrees of freedom.

For information on setting the convergence criterion, see “Providing A Convergence Criterion” on page 26.

Nodal Statistics

The nodal statistics tell you the nodes where, during a given iteration, the minimum and maximum velocities and elevations occurred, and the associated values. The number of **NODES ACTIVE** indicates the number of nodes flagged “wet” for this iteration.



To find this section in the full results listing file, search for the text “**RESULTS AT**”.

Continuity Checks

These statistics are the last information written to the full results listing file for an iteration, unless wetting and drying is requested. In this section you will find the amount of flow across each check line you have defined. The flows are listed as **TOTAL**, **X FLOW**, **Y FLOW**, and **PERCENT**. The percent field is always with respect to the *first* continuity check line and is important for analyzing any loss or gain of mass at the check line. The value here should ideally be 100% for a converged steady state simulation.



To find this section in the full results listing file, search for the text “**CONTINUITY CHECKS ..**”.

Wetting And Drying Information

If wetting and drying calculations are requested with the DE card, the changes made to the mesh, if any, are listed prior to the start of each time step. This information is listed whenever wet/dry checking is done (after every set of LI iterations, as specified on the DE card).

The information lists any element or node numbers which have been removed from or added to the mesh geometry since the last wet/dry check. Here is a typical example with three quadrilateral elements removed:

```
THE FOLLOWING ELEMENTS HAVE BEEN ELIMINATED
 23   24   145
THE FOLLOWING ELEMENTS HAVE BEEN ADDED
THE FOLLOWING NODES HAVE BEEN ELIMINATED
 118  119  124  125  561  562  563  566  567  568
 572
THE FOLLOWING NODES HAVE BEEN ADDED
```



Note: When an element or node is removed from or added to the mesh, the total number of **ACTIVE NODES** and **ACTIVE SYSTEM EQUATIONS** for this iteration will have changed from that of the previous iteration.



To find this section in the full results listing file, search for the text “**ELIMINATED**”.

Time Step Revisions

When revisions are made during a time step, the input Run Control Parameters in the full results listing file are rewritten. After this listing, the run continues as usual, starting with the Buffer Blocks.



Note: No parameters are explicitly flagged as being revised. You must manually compare with the previous run control parameters to find the differences.



To find this section in the full results listing file, search for the text “**REVISION**”.

Steady State And Dynamic Simulation Nodal Results

This section of the full results listing file contains the latest nodal results of the simulation run. RMA2 will write the nodal results after the final time step has converged, and also at the iteration or time step interval specified on the TR card.

*A “Column of Data” in this context consists of the headings **NODE**, **X-VEL**, **Y-VEL**, **DEPTH**, and **ELEV**.*

The nodal results are listed as three columns of data, each containing node numbers, along with each node’s *x* and *y* velocity components, depth, and water surface elevation. The three columns of data are demarcated by the *single* (sub)column with the heading **NODE**. The table is interpreted by reading down the first column of data to the end, then starting at the top of the second column of data and reading down to the end, then moving to the top of the third column of data.

This section also contains **FLOWS AT NODES FOR 1-D ELEMENTS** and the **TOTAL VOLUME IN STORAGE BY ELEMENT TYPE**, which lists the fluid volume in storage for each material type you have defined.



To find this section in the full results listing file, search for the text “**NODAL RESULTS**”.

Understanding The Summary Listing File

The summary listing is simply a brief synopsis of information for any nodes which you specify on TRN cards (see “Summary Results Listing” on page 40 for details on how to obtain a summary listing). For each node, the summary listing file will provide a table for each time step hour, which contains the

- *x* component of velocity
- *y* component of velocity
- total velocity
- flow
- depth
- elevation

The nodes for which data is requested are provided on TRN cards. A typical TRN card will appear similar to the following example.



Example:

```
.  
.   
TRN    23    330    547    636  
.   
.
```

After a simulation run using this example TRN card, the summary listing file will contain a separate table, as described just above, for each node listed on the card.

Interpreting The Solution

It is much easier to interpret the solution file with a graphical user interface, such as FastTABS, than to analyze a printout or stare at x-y plots at individual nodes. Since the solution file is written in binary form and cannot be readily moved across computer platforms, it is very convenient to have both RMA2 and the graphical user interface resident on the same platform. However utility programs, such as BIN2ASC and ASC2BIN, are available to assist when you need to move solution files across computer platforms.

The primary solution file contains the following types of information for each node in the mesh:

- x component of velocity
- y component of velocity
- Depth
- Water surface elevation
- A flag to mark as wet or dry (NDRY)

The solution file also contains the material type number for each element, which is written as negative if the entire element is dry.

It is advisable to examine contours of velocity magnitude, water surface elevations, and velocity vectors at every time step. The following checklist of items should be addressed.

- Are there signs of irregular or oscillating velocity magnitudes?
- Are there any two delta-X instabilities present?
- Are the directions of the velocity vectors reasonable?
- What is the maximum Froude number?
- Have irregular boundaries occurred (such as when wetting and drying)?
- Are some elements abruptly changing between active and inactive states (wetting and drying)?
- Are the boundary conditions specified correctly?

Advanced Techniques

Using One Dimensional Elements

One dimensional element characteristics are defined using GN and GW cards with the N option; normally done in the geometry file. Refer to the GFGEN geometry manual for more details on creating one dimensional elements in the geometry file.



FastTABS Note: As of this writing, in order to allow FastTABS to recognize one dimensional element widths and side slopes, you must use a GN card with the N option to specify the x and y coordinates and the bottom elevation, and a GW card with the N option to specify the surface width, side storage width, and left and right side slopes.

There are 4 types of one dimensional elements:

- The Basic One Dimensional Element
- Transition Elements
- Junction Elements
- Control Structure Elements

Each of these types has a specific purpose.

Using Basic One Dimensional Elements

The process of incorporating and using one dimensional elements is not a difficult concept, but you may want to consider the pros and cons.

Pros

- One dimensional elements provide a very computationally efficient means to extend the mesh domain in order to move the boundary conditions away from the study area.
- It is the only way to incorporate a control structure into the model.

Cons

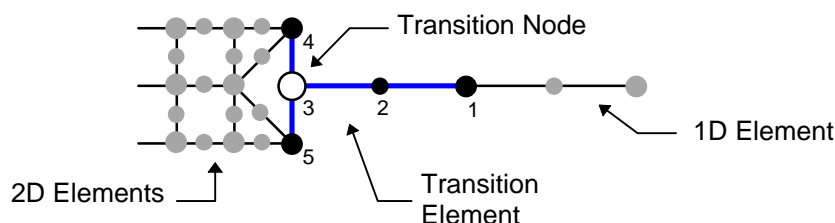
- The TABS sediment model cannot handle one dimensional elements (should be rectified by April 1996)
- At the time of this writing, the graphical user interface, FastTABS, does not permit editing, reordering, contouring, or modifying any

aspect of a one dimensional element (should be rectified by May 1996). This adds a degree of “manual” editing to the modeling process and opens a Pandora's box of potential error.

Using One Dimensional Transition Elements

Transition elements are required to connect one dimensional elements with two dimensional elements.

The T-shaped 5-node element connection table is shown in the figure below. The figure illustrates a suggested technique for transitioning the two dimensional resolution down to one dimensional. Element material type (IMAT) numbers for transition elements may be between 0 and 900. It is important to remember to maintain an angle that is very close to 90 degrees at nodes 4 and 5 in order to prevent “leakage”. A GN card with the N option must be provided for nodes 1, 4, 5, and also for node 3, the Transition node. The bottom elevations of nodes 3, 4, and 5 must be identical in order to conform to the trapezoidal channel assumption. The coordinates of the transition node must be exactly half way between nodes 4 and 5.



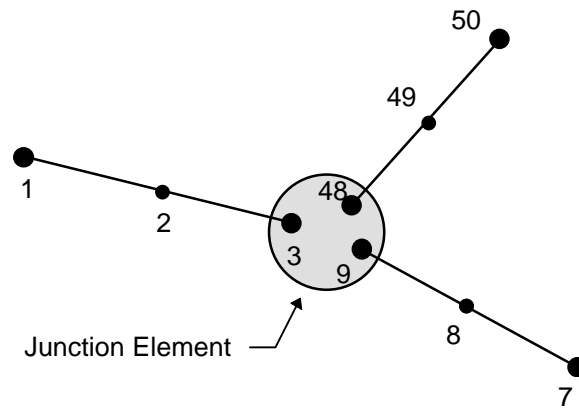
GE	elem#	1	2	3	4	5	0	0	0	imat#

Using One Dimensional Junction Elements

Junction Elements provide a means to connect two or more one dimensional elements together at a central location.

Assume there are n one dimensional elements connected at a junction. At the junction, each element has 2 degrees of freedom; water surface elevation and discharge, for a total of $2n$ degrees of freedom. A well posed mathematical problem will be established if one degree of freedom is “constrained” for each node at the junction. Therefore, we need n equations for the junction to provide these constraints.

In the illustration below, three branching channels are represented by one dimensional elements forming a junction element composed of three nodes: 3, 9 and 48. Each of these nodes may have unique width, side slopes, elevations, and storage widths, but they must all have identical x and y coordinates.



GE elem# 3 9 48 0 0 0 0 0 imat#

Any time three or more one dimensional elements intersect (8 is the maximum), RMA2 needs to be told how to interpret this junction. There are three types of junctions:

- Water Surface Junction (IMAT = 901)
- Total Head Junction (IMAT= 902)
- Momentum Junction (IMAT= 903)

You may control the type of junction by specifying a material type (IMAT) of 901, 902, or 903 for the junction element. The material type is specified on the element's GE card.



Note: The material type (IMAT) for a junction element must be either 901, 902, or 903.

Water Surface Junction

The first type of junction is a “water surface junction” (IMAT = 901). This mandates that mass is conserved, and that all water surface levels will match at the junction.

Total Head Junction

The second type of junction is a “total head junction” (IMAT = 902). This mandates that mass is conserved, and that all total energy heads match at the junction.

Momentum Junction

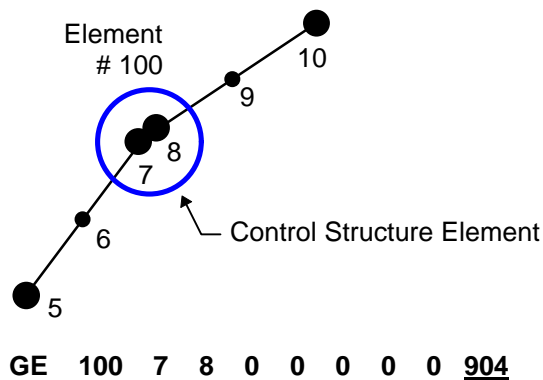
The third type of junction is a “momentum junction” (IMAT = 903). This mandates that mass is conserved. However, the first two nodes in the element connection table of the junction element define the primary channel within which momentum will be conserved.

Using One Dimensional Control Structures

Flow control structures are a means by which you can simulate conditions (weirs, gates, dams, etc.) which were previously impossible to accomplish with RMA2. For this version of RMA2, flow control structures are permitted only on one dimensional sections of the mesh. Work is ongoing to incorporate this feature in a two dimensional network.

Control Structure Element

A control structure element consists of two collocated corner nodes. These two nodes define the structure at two adjoining one dimensional elements. The type of flow to be associated with the control structure element is defined using the FC card. To create a control structure element, the element material type (IMAT) must be greater than or equal to 904. Below is a typical example of a control structure element and the GE card defining it.

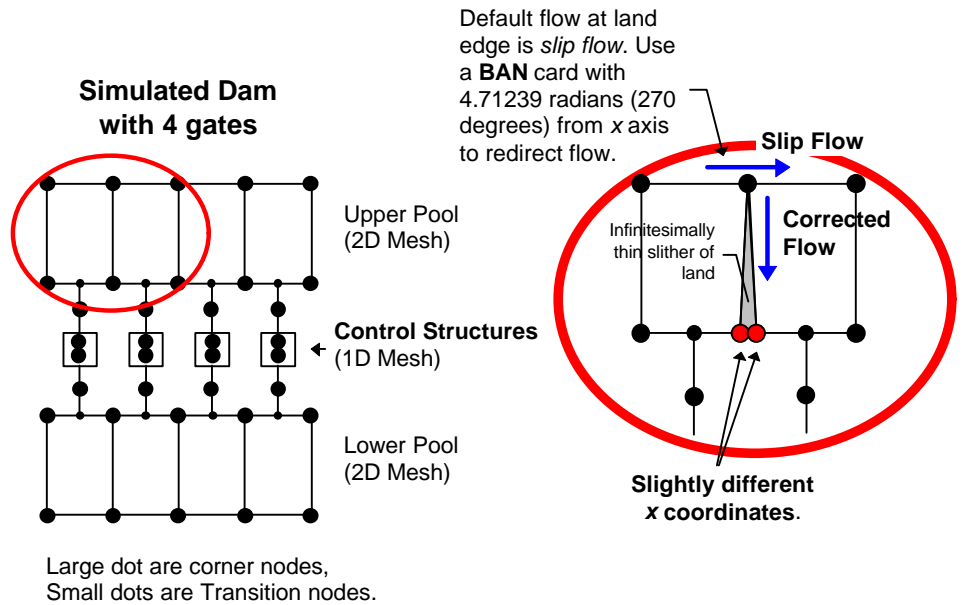


Note: The element connection order for a control structure element defines HN1 and HN2 on the FC card.

Nodes 7 and 8 in the figure are shown at slightly different locations only for the purpose of illustration. To form a proper control structure element, both nodes must be coded at the exact same (x, y) location using GN cards with the N option.

Simulating Spillway Gates

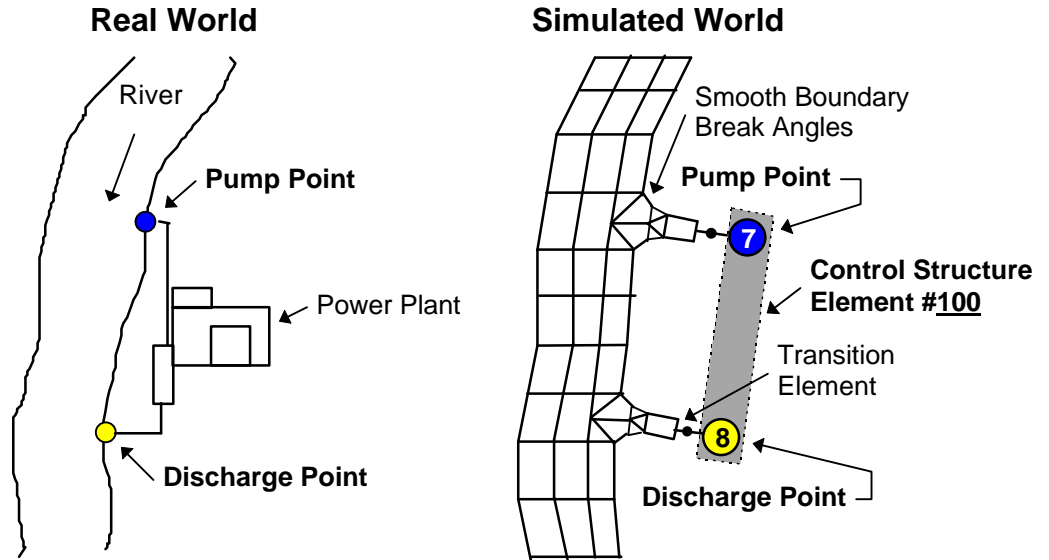
There have been some lessons learned regarding control structures. For instance, if you wish to model several gates composing a dam, as shown in the left figure below, several side-by-side control structures might be used to separate the upper and lower pools. A few advanced modeling techniques must be applied, as indicated by the zoomed figure on the right. To “trick” the model into handling this situation, the x coordinates of the flagged corner nodes are separated by approximately 0.20 ft to create a slight “land” boundary between the two elements. The BA card with the N option is required to set the angle to 4.712 radians for each interior two dimensional corner node approaching and exiting the structure.



Pumping And Discharging Using Control Structures

To form a normal control structure element, both nodes must be coded at the exact same (x, y) location using GN cards with the N option. However, it is theoretically possible to create a “pump” and “discharge” type of scenario if the nodes defining the control structure are placed at different locations. This technique has been applied to a simulation involving a power plant, where water is pumped over a condenser for cooling, then discharged back to the estuary.

To accomplish this, add a control structure element to your mesh, separating the two nodes which constitute the control structure element. Place the first at the *pumping* location, and the second at the *discharge* location. The transition from the two dimensional section to the one dimensional section containing the control structure must be made following the rules for combining two dimensional and one dimensional elements, as described in “Using One Dimensional Transition Elements” on page 60. Also, the use of smooth boundary break angles will help prevent leakage.



In GFGEN geometry...
 GE 100 7 8 0 0 0 0 0 0 904

In RMA2 run control...
 FC 904 1 *discharge* 0 0 0 0

For pump and discharge type situations with two dimensional elements, see “Pumping And Discharging” on page 83.

Resuming A Stopped Simulation

Resuming a simulation is known as Hotstarting. In order to Hotstart, you must have specified the creation of a Hotstart file in the previous run. The starting point, or initial conditions, for the Hotstart will be non-zero velocities and a sloping water surface elevation for all active elements.

The Hotstart file is rewound and derivatives are saved at the end of each iteration that has an acceptable depth convergence criterion. Therefore the last converged iteration is saved to the Hotstart output file.

Why Hotstart?

There are several reasons you may wish to Hotstart RMA2. Here are a few.

- Remove Effects Of The Initial Conditions Eliminate model spin-up, then continue
- Data Security Provide a safeguard/backup in case of machine failure, etc.
- CPU Availability You may have a limited amount of CPU time available for each job run

How To Hotstart

Hotstarting consists of two procedures:

- Creating A Hotstart File
- Restarting Using An Existing Hotstart File

You may Hotstart as many times as you like. Remember that in order to Hotstart, you must have saved a Hotstart file from the previous run.

Creating A Hotstart File

Saving a Hotstart file is simple. You need only to specify a positive value for NLL on the \$L card. When you run RMA2, you will be prompted to enter a name for your Hotstart output file.

Restarting Using An Existing Hotstart File

Here is the procedure:

1. Specify that the initial conditions are to be read from an existing Hotstart file (\$L card, NB = any positive number).
2. If refinement of the previous solution is desired, set the iteration counter, NITI on the TI card, to a non-zero value. RMA2 will compute this many additional iterations toward the previous solution.
3. Specify the number of boundary condition sets (END cards) to skip in the run control file (TZ card, NSTART). Typically set NSTART equal to the last time step which was saved to the Hotstart file you are using as input, **plus one**.
4. Specify mid-iteration switch (TZ card, MBAND).
MBAND is the logic control variable that directs the program to either skip to the next time step (MBAND=0) or to remain at the last time step of the previous run (MBAND=1) to make additional iterations at the final time step of the prior run.

An example is provided below which illustrates the necessary check points you should examine to assure that the Hotstart initial conditions have been read in correctly, and that the correct boundary condition set is active as the model resumes the simulation. The convergence parameters and active nodal statistics from the last time step (in this example, hour 120, time step 5) of the Coldstart run are circled in the first box below. As highlighted in the second box, you should modify the \$L, TI, TR, and TZ cards in the RMA2 run control file to prepare the restart parameters.

As a check point in the restarted simulation, verify that the active nodal statistics are the same as the in the previous run. One last check is to verify that the boundary condition set for the first time step in the Hotstart is valid.

Hotstart Example:

Previous Run

```
RESULTS AT THE END OF    5 TIME STEPS...
TOTAL TIME = 120.000000 HOURS... ITERATION CYCLE = 3

CONVERGENCE PARAMETERS

      DF      AVG  CHG      MAX CHG      LOCATION
      1      0.0001      0.0004      259  X-VEL
      2      0.0001     -0.0003      283  Y-VEL
      3      0.0000     0.0001      309  DEPTH

ACTIVE NODAL STATISTICS FOR THIS ITERATION

      NODE  XVEL-MAX  NODE  XVEL-MIN  NODE  YVEL-MAX  NODE  YVEL-MIN
      354   10.952   164  -10.021   158    1.850    52   -12.285

      NODE  ELEV-MAX  NODE  ELEV-MIN  AVE-ELEV  NODES ACTIVE
      326   121.356    18   110.845   116.797    932
```

Modification To Run
Control for Hotstarting

```
T3 Vital information for Hotstart run control
$L 63 62 60 64 0 3 59
TR 2 -99 1 0
CO NSTART=6 for HOTSTART
TZ 24 720 999 6 0
TI 0 5 .0001 .001
BQL 1 500000 6.17
BHL 2 111
1  END                      Steady State hr=0
2  BQL 1 535000 6.17          hr = 24
  END
3  BQL 1 575000 6.17          hr = 48
  END
4  BQL 1 575000 6.17          hr = 72
  END
5  BQL 1 600000 6.17          hr = 96
  END
6  BQL 1 650000 6.17          hr = 120  <-- Previous run stopped here
  END
7  BQL 1 700000 6.17          hr = 144
  END
8  BQL 1 750000 6.17          hr = 168
  END
```

After
Hotstart Run

```
..... INITIAL CONDITIONS .....

RESULTS AT THE END OF    0 TIME STEPS...
TOTAL TIME = 120.000000 HOURS... ITERATION CYCLE = 3

CONVERGENCE PARAMETERS

      DF      AVG  CHG      MAX CHG      LOCATION
      1      0.0000      0.0000      0  X-VEL
      2      0.0000      0.0000      0  Y-VEL
      3      0.0000      0.0000      0  DEPTH

ACTIVE NODAL STATISTICS FOR THIS ITERATION

      NODE  XVEL-MAX  NODE  XVEL-MIN  NODE  YVEL-MAX  NODE  YVEL-MIN
      354   10.952   164  -10.021   158    1.850    52   -12.285

      NODE  ELEV-MAX  NODE  ELEV-MIN  AVE-ELEV  NODES ACTIVE
      326   121.356    18   110.845   116.797    932

*** BOUNDARY CONDITIONS DEFINED FOR TIME = 144.0000 HRS
BQL 1 700000 6.17
```

Common Causes For Hotstart Failure

Invalid Initial Conditions

The initial condition read does not match the specified starting boundary condition. That is, the value of NSTART on the TZ card is wrong.

Mesh Modifications

The mesh has been modified between runs. Even a minor depth change is enough to cause divergence.

Specifying Units

The default for RMA2 is English units. However, if you wish to operate in metric (SI) units, the following is required:

- Use an SI card to specify metric units and position it immediately after the \$L card in the GFGEN geometry file.
- Run GFGEN to create a new binary geometry file containing data in SI units to be used as input for RMA2.
- Use an SI card in the RMA2 run control file (boundary condition file) instructing RMA2 to use SI units.
- Provide all mesh revisions (GN card, GS card, GZ card, and GW card, etc.) in meters.
- Non-default values for wetting and drying on the DE card and DM card must be entered in meters.
- Automatic roughness by depth (RD card) must reference depth in meters.
- Turbulent Exchange coefficients (eddy viscosities) must be entered in SI units (Pascal-sec) if using an EX card, EY card, or EV card.
- Non-default fluid density values must be entered in SI units (kg/m^3) on the FD card.
- Use SI units for Coldstart initial conditions on the IC card.
- Use SI units for *all* boundary condition specifications.

Customizing The Solution File

RMA2 provides ways to customize the solution file. These include the TO card and the Hotstart technique.

Throwing Out Initial Condition Contamination

There may be times when you do not want to save any part of the solution that may have been contaminated by the initial conditions (spin-up). There are two basic ways to exclude this time period from the solution file:

- Run the simulation through spin-up, then Hotstart (see “Resuming A Stopped Simulation” on page 64).

- Use the TO card to specify the decimal hours you want to save. Only the specified time periods will be written to the solution file.



Note: The **MERGAVG** program is a utility available which can assist with post processing the solution. See “Utilities” on page 195 for details.

Customizing The Full Results Listing File

There are several options available for controlling the information that is written to the full results listing file. Customizing gives you control over the types of data which are written to the listing file, and how often. The TR card is used to specify what will be in the full listing.

When To Write To The Full Listing File

The full listing can become huge if not properly controlled. To reduce the file size you may request that data be written to the full listing either on an iteration basis, or a time step basis. This is done by assigning a value to ITSI on the TR card as follows:

- Positive values for writing at *iterations*
- Negative values for writing at *time steps*

The value you specify is used as the interval at which the data will be written. For example, if you specify a -3, the results and associated data at every third *time step* will be written to the listing file.



Tip: If disk space becomes limited, change the value of ITSI to -12, or some other larger negative number (such as -99). This will cause RMA2 to only write data for time steps at this specified interval (except for wetting and drying). Alternatively, to prevent writing a full listing file altogether, you might set the value of IOUW on the \$L card to zero, and enter “null” at the results listing filename prompt.

Including Startup Conditions

There are several types of startup condition data which can be included in the full results file. These are:

1. Echo of the run control data
2. Geometry information
3. Wetting and Drying/Marsh Porosity information
4. Elemental area information
5. Nodal specifications defined at startup
6. Roughness (Manning’s n-value) assignments
7. Front width size and total equations

Suppressing Node And Element Data

If you are confident about your geometry data and you want to reduce the size of the full listing file, you can eliminate node and element data from the listing file. To do

so, set IPRT on the TR card to zero. This will eliminate items 2 through 5 in the above list of startup conditions.

Suppressing Coldstart Initial Conditions

The convergence parameters written to the listing file for initial conditions at Coldstart are always zero because RMA2 has not yet obtained any solution. Therefore, it is uninformative to see this information. To eliminate the Coldstart initial conditions, set IPRT on the TR card to 1. This will also eliminate the echo of the convergence parameters at startup.

Including The Manning's n -value Information

If you have manually assigned the roughness parameters and are confident of those assignments, then you may not wish to save this information in the full results listing. To eliminate the table, set IPRT on the TR card to 3.

Including Your Input Data In The Full Listing File

You may include your input run control data in the full results listing file. Having this information available in the listing file may assist you when locating problems.

To include an echo of the run control input data, set IECHO on the TR card to 1.

Requesting Only A Summary Of Nodal Results

When you only need a summary of the simulation results at specific nodes, you can request a summary listing file using the ISPRT parameter on the \$L card. The nodes included in the summary listing file are specified using the TRN card. The summary results listing file contains

- A table of the steady state solutions of x and y velocity components, depth, etc., for each node listed on the TRN cards
- A dynamic hydrograph of the above information for each node listed on TRN cards



Tip: Information in the summary listing can be imported into a spreadsheet program for plotting and further analysis.



Note: None of the parameters on the TR card will affect the summary listing. Data is written to the summary listing file after every time step, and there is no way to change the type of data the file will contain.

Plugging In Mesh Modifications

Normally, mesh modifications are made prior to the RMA2 run, and no mesh modifications are necessary at the time of the run. Occasionally it is desirable to create or modify an existing mesh from within the RMA2 run control file. This ability allows for quick testing without having to go back through GFGEN.



FastTABS Note: FastTABS will not recognize mesh modifications made from within the run control file. If you change the number or location of nodes or elements in the run control file, the solution file you obtain after the RMA2 simulation run will no longer match the original geometry file.

Modifying Nodes

Nodes can be added to the mesh, or moved, by using a GN card in the run control file.

Adding Nodes

Any node which is added to the geometry must be connected to the mesh via an element. Nodes are typically added only when adding a new element.

Corner Nodes

To add a corner node, modify the run control file and use a GN card with the N option. Assign a node number which is larger than the largest node number currently existing in the mesh, accompanied by the x , y , and z coordinates for the new node. Modify or create the element connection table using a GE card to include the new node in the mesh.

Midside Node

To add a midside node, modify the run control file and use a GE card to describe the element connection table for the element to which the node belongs. Assign a node number which is larger than the largest node number currently existing in the mesh

Moving Nodes

To move an existing node, modify the run control file and use a GN card. Specify the node number of the node which you want to move, along with the new x and y location for the node.

Modifying Elements

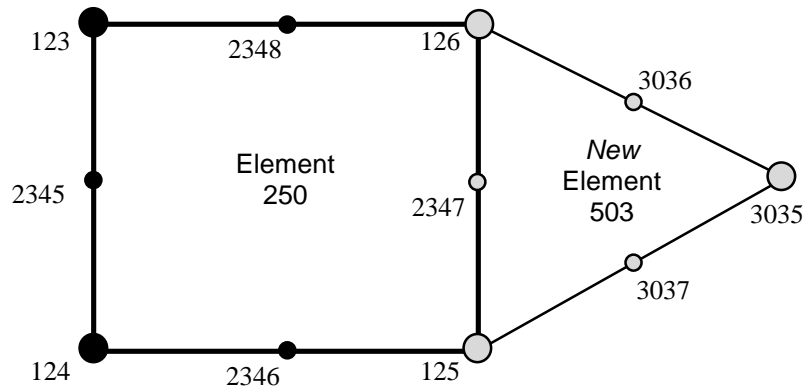
Elements can be added to or removed from the mesh, or the material type changed by modifying the run control file.

Adding Elements

Before adding an element, *all* corner nodes necessary to build the element must have been previously added to the mesh (see Adding Nodes). It is imperative that you know all the corner and midside node numbers of the face of the element to which you are attaching the new element. You will need to include these numbers in the new element's *connection table* (GE card). Start the connection table with a corner node and proceed around the element in a counterclockwise fashion.

The node numbers for the uncoupled midside nodes created for the new element must be unique. You will need to know any node numbers which are unused in the mesh. If there are none, the maximum node number, midside nodes included, of all the nodes in the mesh will be your starting point for numbering the new element.

Prior to modification,
The largest *element* number is 502.
The largest *node* number is 3034.



Example:

```
CO Original element (Quad element)
GE 250 123 2345 124 2346 125 2347 126 2348 1 0.0
.
.
CO New element (Triangular element)
GE 503 125 3037 3035 3036 126 2347 0 0 1 0.0
```

To add an element, insert a GE card in the run control file. Specify an element number not previously defined in the mesh, along with the list of nodes in a counterclockwise direction, and a material type.

Removing Elements

To remove an element, simply assign a material type of zero for the element. This can be accomplished by using the GT card, or alternatively with the GE card, to set the material type.



Note: Removing elements by setting the material type to zero does not *physically* change the geometry, but effectively removes the element by converting it to land. The element can be re-activated at any time by changing the material type to a value other than zero.

Changing An Element's Material Type

Normally, an element's material type is set in the GFGEN input geometry file. You can change the material type for the RMA2 run for an element or elements by using a GT card. Specify the element number along with the material type number you wish to assign.



Note: Any new material type numbers must have roughness and turbulent exchange coefficients defined, typically accomplished using an EV card.



Tip: To effectively remove an element from the computational mesh, set the material type (IMAT) for the element to a value of zero (the element will appear to be land).

Curving Element Edges

There are several reason why you may want to curve elements.

- Add aesthetics
- Add length to a section without adding resolution
- Represent meandering channels
- Reduce boundary break angles without adding additional resolution

Curving can be useful to help improve mass conservation and to achieve a more natural appearance along the mesh boundary. You may curve both one dimensional and two dimensional elements.

To curve an element side, add a GN card to the run control file, specifying the new x and y location for the midside node that lies on the element edge which is to be curved. The bottom elevation for a midside node is ignored because it is calculated from the elevation values of the two adjacent corners.



Note: Caution is warranted when curving interior element edges.



Warning: Be aware that curving element edges may result in an ill-formed element when the new location chosen for the midside node is too far away from where it would be normally.

Changing Bottom Elevations

Within the RMA2 run control file, the bottom elevation may be changed on the GZ card, or the GN card with the N option.



Warning: The GN card with the N option also specifies the x and y coordinates, as well as the bottom elevation, z .

Resizing The Mesh

RMA2 provides the capability to resize your mesh by applying scale factors. All x , y , and z coordinates of the mesh are multiplied by the XSCALE, YSCALE, and ZSCALE factors located on the GS card. To resize the mesh, specify values other than 1 for the x , y , and/or z scale factors. This technique has been used to convert from English to metric, or to double the size of a study area for the sake of sensitivity.

Changing An Element's Eddy Viscosity Tensor

The direction of the eddy viscosity tensor for a given element (radians, counterclockwise from the $+x$ axis) may be modified by using the GV card, or alternatively, the GE card.

Advanced Boundary Condition Techniques

Revising boundary conditions can help start the run without divergence by wetting the entire grid and gradually obtaining the actual water surface through revisions.

Revising Boundary Conditions During A Simulation

The boundary conditions which were defined during the initial time step of the simulation will hold until you specifically reference them to change. A new boundary condition may be assigned in the middle of a time step after a REV card, or at the end of a time step after an END card.

The REV card allows you to revise the run control in the middle of a time step. The revision may reference the boundary conditions, time or print controls, roughness, eddy viscosities, wet/dry parameters, etc. However an error will result if you attempt to revise any boundary condition that has not been initially defined.



Note: Geometry modifications other than modifying the material type will likely result in a diverged solution.



Example:

```
T3
$M
$L
.
.
.
EV 1 500 500 500 500 0.03
TI 4 0 .01
.
.
BQL 6 50000 -1.2
BHL 7 100
Solve for Hour 0.0 REV Now lower coefficients and increase Q for 2 iterations
TI 2 0 .01
EV 1 300 300 300 300 0.027
BQL 6 75000 -1.2
Solve for Hour 0.0 given new data REV Now lower coefficients more and increase Q
EV 1 150 150 150 150 0.025
BQL 6 100000 -1.2
Solves for Hour 0.0 and writes the solution using
EV = 150 and Q = 100000. END Marks the end of the time step
```

Changing The Direction Of Flow

All exterior boundary nodes that touch a land edge are automatically assigned an appropriate flow angle. Nodes at flow boundary condition locations, however, should be manually assigned flow angles which indicate the true direction of flow. The angle may be specified on the BA card, BQ card, BRA card, or BRC card.



Tip: Occasionally, you may need to force a flow angle for a particular node to be a given direction, such as when using side by side Control Structure Elements (i.e., multiple flood gates, etc.). A BA card may be used to directly specify the flow angle.

Changing The Speed Of The Current

An alternative to the conventional means of specifying the flow conditions at a boundary is the current speed/direction option. The boundary current speed, or velocity magnitude, is defined on the BS card. If a BS card is used, a BA card must also be specified to define the flow angle.



Note: The current speed/direction boundary condition is independent of the water surface, therefore the total flow across the nodes using a BS and BA card will fluctuate with the tidal signal.

Boundary Permeability (Reflection/Absorption)

The reflection/absorption boundary condition is useful whenever a boundary condition cannot be extended to the head of tide or null point. This boundary condition has been found to work far better when applied to a one dimensional element rather than to a two dimensional element.

The reflection/absorption boundary condition (BRA card) is a special form of the rating curve (BRC card). It calculates the discharge based upon a calculated water surface elevation and the surface area of tidal storage beyond the boundary. The variable BETA on the BRA card is a decimal number which may vary between 0.0 for total absorption to 1.0 for total reflection of the tidal wave.

Hotstarting using this boundary condition is a little unusual, in that the steady state water surface value (SSWSE) for the first node on the continuity check line referenced by the BRA card must be supplied to at least 4 places past the decimal. The variable SSWSE is used as the mean value about which the wave oscillates.

Varying Discharge During A Simulation (Rating Curve)

Most rivers have a boundary rating curve defined from historical data. The BRC card allows you to supply the coefficients of an equation to provide a fit to that historical data. Once the equation is defined, RMA2 will compute the boundary discharge based upon the calculated water level for that time step.

Bendway Correction (Vorticity)

Traditional RMA2 depth averaged calculations of flow around a bend frequently over predicted streamwise velocities on the inside bank of a river. When water flows around a corner, a radial acceleration is developed that forces the surface water to the outside of the curve and the water near the bed to the inside of the curve. This is the familiar “secondary” or “helical” flow pattern which causes sediment aggradation on the inside of curves and soil erosion on the outside of curves. RMA2 cannot adequately predict the effect of this behavior on the depth-averaged velocities. In order to improve predictions of depth-averaged velocities around curves, a secondary flow corrector, coined as bendway correction, option was added to RMA2.

The Principle Of Bendway Correction

The theoretical basis of the bendway correction was developed by Dr. Bernard of WES-HL for his depth averaged finite difference numerical model, STREMR. For additional information, reference US Army Engineers technical report HL-92-9, Bernard, R. S., and Schneider, M. L. (1992), "Depth-Averaged Numerical Modeling for Curved Channels".

The bendway correction is accomplished by first solving an additional equation for the transport of streamwise vorticity. Vorticity is a measure of rotation of flow. Streamwise vorticity at a point is equal to the velocity of the fluid about the axis in the streamwise direction of flow. Streamwise vorticity is in the vertical plane perpendicular to the direction of flow and is related to the radial accelerations that cause the helical flow pattern.

The transport equation for streamwise vorticity is

$$\frac{\partial \Omega}{\partial t} + \frac{\partial \Omega}{\partial x} + \frac{\partial \Omega}{\partial y} = \frac{A_s \sqrt{C_f} |u|^2}{Rh(1 + 9h^2 / R^2)} - D_s \sqrt{C_f} \Omega \frac{|u|}{h} + \frac{1}{h} \nabla(nh \nabla \Omega)$$

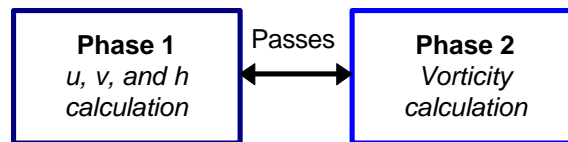
where Ω is the streamwise vorticity, $|u|$ is the magnitude of the velocity vector, R is the local radius of curvature, C is the friction coefficient, h is the depth, $A_s = 5.0$, and $D_s = 0.5$. The units of vorticity are sec^{-1} .

The additional shear stress caused by the secondary, helical flow is calculated from streamwise vorticity at each node. The components of this shear stress are added to the other terms (friction, slope, Coriolis) which occur in the governing equations.

The Bendway Correction Solution Scheme In RMA2

The vorticity calculation in RMA2 is a two phase process: the traditional u , v , h phase, and the vorticity phase.

The addition of the bendway correction required a number of changes to RMA2. In order to minimize computer storage requirements, it was decided to calculate streamwise vorticity separately from velocities and depth. The calculation of vorticity (phase 2) begins after the calculation of velocities and depth (phase 1) has converged. The relationship between the two phases is shown in the figure below.



Velocity and depth are held constant during the vorticity calculations, and vorticity is held constant during velocity/depth calculations. Once the vorticity calculation has converged, the program returns to the velocity/depth calculations. As is true for the velocity/depth phase, the computation of vorticity is done iteratively. In iterative calculations, the solution is approached in a series of "correcting" steps. If either phase converges to a solution before the maximum allowable number of "correcting" iterations, then this is considered to be a "good pass". A vorticity calculation is considered to have converged if the corrections being made are less than the value of the variables SSVCRT or USDVCC on the VO card. The maximum number of iterations allowed for the vorticity calculation is set by the variables NVITI or NVITN on the TV card. If the number of passes between the phases reaches NPASS1 or NPASS2, then the calculation has failed to converge. If the number of "good passes" between the two phases exceeds NGOODMAX, the calculation is finished and the code can proceed to the next time step.

How To Apply Bendway Correction

The bendway correction option is deactivated by default. Under normal conditions, the bendway correction will require the \$L card, BV card, TS card, TV card, and VO card.

Vorticity Boundary Condition

The vorticity calculations require that a vorticity boundary condition be specified. The BV card with the L option is used to apply the boundary condition for the vorticity calculation. Only one boundary condition for vorticity is used, and this will normally be at the inlet of the flow problem. Provide variable J1 with the continuity line number at the desired boundary, and the variable VOR with the value of vorticity along this line. VOR will usually be set to zero.

Iteration Control For Vorticity Calculations

Variables on the TV card control the maximum number of iterations of the vorticity phase, and the maximum number of passes between phases (see “The Bendway Correction Solution Scheme In RMA2” on page 75 for a discussion of the vorticity phases). For steady flow problems, NPASS1 sets the maximum number of passes, and NVITI controls the maximum number of vorticity iterations. For unsteady flow problems, these variables are NPASS2 and NVITN, respectively. For either steady or unsteady flow problems, NGOODMAX sets the minimum number of “good passes” required before the calculation is stopped; i.e., when both phases are converging well for NGOODMAX number of times.

Setting Up The Vorticity Calculation

Parameters on the VO card enable the bendway correction, set the convergence criteria for the vorticity calculation, and provide the constants for the vorticity transport equation. If the bendway correction is desired, enter the number 1 for the variable IVOR. A small positive number (.00001 is the default) should be entered for SSVCRT if it is a steady flow problem, or for USDVCC for unsteady flow problems. As the vorticity calculation proceeds, changes are being made to the current estimate of vorticity at each node. The magnitude of these changes is referred to as the “norm”. After every iteration, the norm is compared to SSVCRT or USDVCC. If the norm is less than SSVCRT or USDVCC, then the vorticity calculation is considered to have converged, and the vorticity calculation finishes before the number of iterations has reached NVITI or NVITN. The last three variables on the VO card should be entered as follows: ASEC = 5.0, DSEC = 0.5, and RCMIN = 6 feet (or 2 meters).

Even if the number of good passes exceeds NGOODMAX, and the calculation stops, you are cautioned to examine the convergence of the individual phases before concluding that the whole calculation has converged. In general, each subsequent pass to the vorticity phase should require fewer iterations before the norm reaches SSVCRT or USDVCC. Similar behavior should be seen during the (**u**, **v**, **h**) phase. Also, the norm for the first iteration of a phase should decrease as the calculation proceeds. Indeed, in a satisfactory calculation, the norm of the *first* iteration of each phase should approach the value of SSVCRT or USDVCC before the whole calculation is considered to have converged.



Tip: Although experience dealing with vorticity in RMA2 is limited at the time of this writing, the technique of converging the traditional (**u**, **v**, **h**) phase of

the RMA2 simulation first, then applying a REV card to activate the vorticity calculation has proved successful. This brings the solution to the correct boundary conditions, eddy viscosities, etc. before applying vorticity calculations to the model.

Obtaining Vorticity Results

RMA2 permits the results of the vorticity calculations to be saved to a unique file with an RMA4 binary solution format. If an output file of vorticity results is desired, a non-zero value must be supplied for NOPTV on the \$L card and for IBVO on the TS card. Enter a positive integer for NOPTV and a positive or negative integer for IBVO, depending on the desired frequency of results. These results are useful only from a diagnostic viewpoint.

The effects of vorticity on the velocity are incorporated within the traditional (**u**, **v**, **h**) RMA2 solution file.

Fluid Properties

Fluid Density

RMA2 provides the capability to alter the fluid density to meet a certain physical condition. The fluid density may be modified on the FD card. The default density value is that for fresh water: 1.935 slugs/ft³.



Note: RMA2 has been found to be indifferent as to whether you specify the density of fresh water or salt water.

Fluid Temperature

RMA2 provides the capability to alter the fluid temperature to meet a certain physical condition, or perhaps a season of the year. The temperature may be modified on the FT card. The default temperature value is 15 degrees Celsius.



Note: Differences in results between model runs when making minor changes to the temperature value have been found to be insignificant.

Automatic Friction Assignment

For most geometry settings, the automatic friction assignment is a very powerful tool, and permits a realistic means by which to model bottom roughness. By using the RD card, you can specify coefficients to define an exponential curve which will be used after each iteration to re-calculate the Manning's roughness coefficient for every element. The depth at each Gauss point of the element is used as the depth for the friction equation.



Tip: Remember that RMA2 does not account for wall friction. You may want to compensate by assigning the elements along the "wall" a unique material type.

The RD card with the T option will permit you to exaggerate the roughness in that area.

Automatic Turbulence Assignment

For most geometry settings, the automatic assignment of elemental turbulence coefficients (eddy viscosity) by Peclet value is a very powerful tool. By using the PE card, you can provide *real time* adjustment of eddy viscosity based upon the computed velocity and individual size of each element.



Note: The velocity used for the Coldstart variable VPEC must be sufficiently large as to permit a successful first guess with the automatic feature.

Generally, larger elements, and elements with higher velocities will have larger eddy viscosity values. Smaller Peclet numbers will result in larger values of eddy viscosity. A recommended range of Peclet values is between 15 and 40.



Tip: Experience has shown that most simulations are more stable if the Coldstart condition begins using the traditional eddy viscosity assignment made with the EV card, then revising (REV card) the run control file to allow for automatic assignment of elemental turbulence (PE card).



Example:

```
T3
$M
$L
.
.
.
EV 1 500 500 500 500 0.03
TI 8 0 .01
.
.
BQL 6 50000 -1.2
BHL 7 100
REV Over-rule the explicit eddy viscosity with automatic Peclet=20
PE 1 20 1.0 1 1 1 1
REV Lower coefficients more by specifying the automatic Peclet=40
PE 1 40 1.0 1 1 1 1
END Marks the end of the time step
```

Solve for Hour 0.0

Solve for Hour 0.0 given new data

Solves for Hour 0.0 and writes the solution using Automatic Peclet = 40.

Wetting And Drying

In the real world, there may be times when portions of an area may become devoid of water, and later become wet again. This process of flooding and draining is referred to as “wetting and drying”. RMA2 provides two means by which to

simulate these events; elemental elimination (DE card) and marsh porosity (DM card). Either of these methods may be used alone or in combination.

For details on the wetting and drying methods in RMA2, reference the following:

Roig, Lisa C., (1995) "Mathematical Theory and Numerical Methods for the Modeling of Wetland Hydraulics," in Water Resources Engineering, Proceedings of the 1995 First International Conference, San Antonio, Texas, August 14 - 18, 1995, American Society of Civil Engineers, New York, pp. 249 - 253.

Elemental Elimination

When used alone, the wetting and drying by element technique eliminates or dries the element when any *one* node on the element has a depth less than or equal to the value of DSET on the DE card. Once an element "dries", flow must go around the newly formed land boundary until the projected water depth calculated by RMA2 exceeds the value of DSETD (DE card) for *all* nodes of the element. To prevent an element from oscillating between wet and dry states, it is recommended to make $DSETD > DSET$.

The mesh should be constructed such that the edges of the elements fall on smooth contour lines such that the storage area wetted or dried at any given moment is minimized. Mesh detailing should be concentrated in the transition regions between wet and dry states. Numerical instabilities can be introduced if an irregular "saw tooth" shoreline results from the newly formed dry land boundary, or if a large storage area is abruptly gained or loss.

Ponds

If the prototype bathymetry of the system dictates the appearance of pond areas, you may have to alter the bathymetry to achieve a monotonically sloping bed to effectively remove the ponds from the simulation. If an isolated pond forms, RMA2 is likely to diverge when the pond is reattached to the wetted network.

Frequency Of Wet/Dry Checking For Elemental Elimination

If a DE card is present, a wet/dry check is always made at the end of a time step. The value of LI on the DE card controls the frequency of checking for wetting and drying by element within a time step. If LI is set too small, the model will not be allowed to converge to a solution based upon the newly formed land boundary before it attempts to proceed to the next wet/dry check. It is safer to set LI to a multiple of the iteration counter (TI card) or greater and let the criterion for satisfactory depth convergence determine when the solution is converged. Hence the values of LI on the DE card must work in harmony with the iteration counter NITI, NITN, SSDCRT and USDCRIT on the TI card.

Advantages Of The Elemental Elimination Method

When the mesh is properly constructed for this wet/dry technique, the RMA2 model will run faster because elemental elimination reduces the number of equations to solve, and the simulation will tend to have a visually realistic wetting and drying appearance.

Disadvantages Of The Elemental Elimination Method

The values of LI, DSET, and DSETD on the DE card are constant for the entire mesh. There is a greater potential of numerical divergence because of the following conditions:

- Irregular boundary shorelines with sharp corners are created if element edges do not align with smooth bathymetric contours.
- Large areas becoming wet or dry during a time step can cause abrupt changes which induce numerical shocks in the simulation.
- Fragmented mesh domains can be created (i.e., potential isolation of mesh regions also known as “puddles” or “ponds”).

Typical Problems Encountered While Using Elemental Elimination

When using the DE card, non converging water depth or velocity parameters usually indicate that the simulation is unstable. When the maximum change convergence parameter is greater than 10.0 at the end of any time step, the mesh is probably diverging. Typical causes of this non-convergence include the following:

Problem 1: Oscillations in water surface elevation or velocity

As the water surface drops and an element or series of elements is marked dry, the mass contained in these elements is shifted to the adjacent elements. If the volume of water in these drying elements is relatively large, it can cause the water surface elevation of the adjacent elements to increase enough to re-wet the just dried elements causing oscillations in both water surface elevation and velocity.

Solution

Make elements smaller in areas that are wetting and drying. Also, make sure that the elevation difference on any element that is wetting and drying is not too great. This will minimize the volume of water that is being transferred to a wet element. Another solution is to use the marsh porosity technique (DM card).

Problem 2: Non-smooth interior boundaries

Most users spend a great deal of time making sure the exterior boundary is smooth. However, when a row of elements on the exterior of the mesh becomes dry, the mesh boundary becomes the edge interface between the outside row of elements and the next row toward the interior. If this new boundary is not smooth then all of the problems inherent to a rough boundary will be experienced.

Solution

Move or create new nodes so that the interior element edges are parallel to the contours.

Marsh Porosity

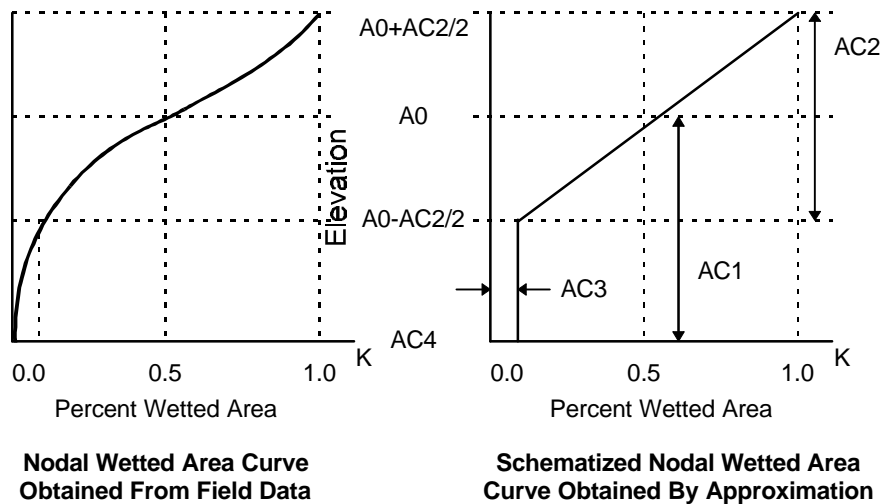
The “marsh porosity” tool in RMA2 allows elements to transition gradually between wet and dry states. This technique allows RMA2 to lower the ability of the element

to hold water; like squeezing a sponge. It improves calculation of the shoreline boundary position for intermittently flooded wetlands and tidal flats.

The residual water volume existing on a partially wet element is calculated by vertically integrating a wetted area curve associated with each node of the element. The wetted area curve represents sub-grid scale bathymetry variations that occur in the area surrounding the node. Partially wet elements are retained until all associated nodes become dry. Dry elements re-enter the computations as soon as one associated node is re-wet.

Marsh porosity is either *on* or *off* for all elements in the mesh during the simulation. However, if marsh porosity is on, and all nodal depths for a particular element exceed the transition range of the distribution (i.e., completely wet element), then the element will carry 100% of its capacity, and marsh porosity has no effect within that element.

The wetted area curve defines the surface area of standing water as a function of water surface elevation. This curve can be determined empirically if detailed contour maps or aerial photos are available. An example of an empirical wetted area curve is shown on the left in the figure below. Note that the wetted surface area, K , has been normalized to the total nodal area such that $0 \leq K \leq 1$. The data required to construct a wetted area curve is frequently unavailable, so that the user must supply an approximate function to describe the wetted area curve for each node. One such approximate function is illustrated on the right. This schematized wetted area curve has the benefit of being easy to integrate, and can be completely described with four parameters: AC1, AC2, AC3, and AC4.



The effective wetted surface area will continue to decrease as the depth of additional nodes on the drying element fall below the transition range. If you suspect that all nodes on an element will have the potential to be flagged dry, then the DE card is required to accompany the DM card to allow for proper removal, and later reinsertion, of the element. If *all* nodes on the element have a calculated depth less than DSET on the DE card, then the element is considered to be completely dry and is eliminated from the simulation. Although the element may be eliminated and a newly formed land edge is created, the Marsh Porosity option has specified that the storage of the element has been gradually reduced, and the likelihood of numerical shocks is drastically reduced. Dry elements re-enter the computations as soon as *one* of the nodes on that element has a depth greater than DSETD on the DE card.



Note: The Marsh Porosity rules for completely eliminating dry elements from the network and reinserting rewet elements are different than with the Elemental Elimination method.

The data required to determine the parameters for marsh porosity are typically unavailable, so you must approximate. Care must be taken to have similar marsh porosity parameters between adjacent nodes.

Here are some additional guidelines for selecting the values of AC1, AC2, and AC3 on the DM card.

- A value of AC1 too small will cause the node to dry prematurely and flow volume will be underestimated.
- A small AC2 makes the distribution behave like a step function which will result in abrupt wet/dry changes similar to a narrow feeder channel opening abruptly onto a wide flat marsh plain.
- AC3 must be small (on the order of 0.01) but always greater than zero.

Many of the same mesh construction guidelines for elemental wetting and drying should be followed when using marsh porosity.

Frequency Of Wet/Dry Checking For Marsh Porosity

When the Marsh Porosity is on, RMA2 automatically calculates the effective nodal depth at every iteration, but if a DE card is present at the same time, the LI parameter controls the frequency of checking for when elements need to be removed from or reinserted into the mesh.

Advantages Of The Marsh Porosity Method

- The mesh design can be more lenient.
- You have total control of parameter assignments.
- The solution will be more stable, and tidal storage is appropriately handled.

Disadvantages Of The Marsh Porosity Method

At the time of this writing, FastTABS will display partially dry marsh elements as all wet even though the water surface elevation may be below the bed elevation. Therefore, the FastTABS user is encouraged to examine *depth contours* rather than *water surface elevation* values.

Typical Problems Encountered While Using Marsh Porosity

Problem 1:

Instability due to different values for marsh porosity parameters being located side by side in the mesh.

Solution

Choose the marsh porosity parameters so that there is not a sudden change between adjacent areas. Select values so that there is a somewhat gradual change between areas.

Problem 2:

Instability due to the selection of AC1 on the DM card.

Solution

The first guess for the appropriate value of AC1 should be the difference in cross sectional elevation along the edge of the element. If instability persists, then try gradually increasing the value of AC1.

Element Inflow And Outflow

RMA2 provides a means to add fluid to or remove fluid from an element or multiple elements. This ability can be utilized to simulate events such as pumping and discharging, or rainfall and evaporation. The way of thinking about these two situations may differ slightly, but the process is the same.

Pumping And Discharging via Element Flow

Simulating a pumping and/or discharge event can be accomplished by specifying an outflow (pump) and/or inflow (discharge) boundary condition at an element. This technique is used by adding fluid to or subtracting fluid from the specified element. For a two dimensional element, the outflow or inflow is measured per unit area (ft^2 or m^2) of the element, whereas for a one dimensional element, the outflow or inflow is measured per unit length (ft or m) of the element.

RMA2 expects a value for *inflow* only. To provide an outflow, you must use a negative value for inflow.

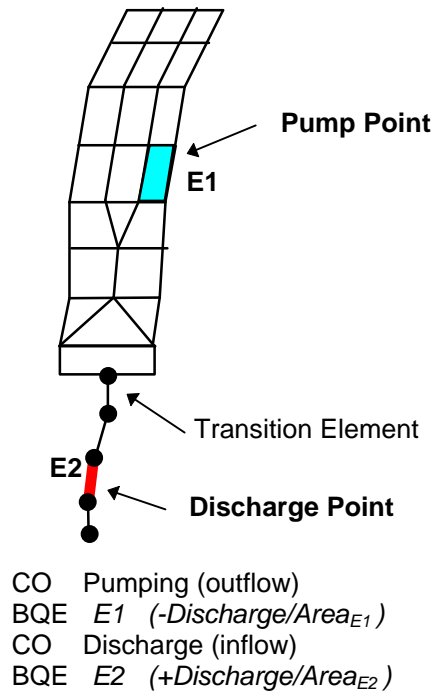


Note: A negative inflow is an outflow.



Example

The figure below illustrates pumping for a two dimensional network and discharging for a one dimensional network. For example, to pump 5000 cubic feet per second (cfs) from a single 2D element, **E1**, with an area of 1000 ft^2 , the value for the elemental inflow per unit area will be *negative* $5.0 \text{ ft}^2/\text{sec}$. To discharge 5000 cubic feet per second (cfs) from a single 1D element, **E2**, with a length of 50 ft, the value for the elemental inflow per unit length will be $100 \text{ ft}^2/\text{sec}$.



For pumping and discharge applications using control structures, see “Pumping And Discharging Using Control Structures” on page 63.

Adding Rainfall And Evaporation

“Rainfall and evaporation” is a special type of boundary condition. Rainfall and evaporation can be applied to a single element, or to many elements in the network. Rainfall or evaporation is applied in inches/hour if using English units, or cm/hour if using Metric units. These values are internally converted to feet/second or meters/second.

The elements you want to influence with rainfall or evaporation can be specified with the RA card using the *by element* option, E, or the *by material type* option, T. All elements in the mesh can be included by using the RA card with no option.



Note: You cannot apply rainfall or evaporation to junction elements or control structure elements, or elements which have a material type (IMAT) greater than 900. When globally assigning rainfall and evaporation, these elements are automatically skipped by the program.

Rainfall and evaporation can occur at any time during the simulation. For example, if you want rainfall to start at hour 30 and stop at hour 33, and evaporation to begin at hour 50 and stop at hour 60, start the simulation with SIDF on the RA card set to zero. At hour 30, update the parameter on the RA card, making it a positive value. At hour 33, update again returning the rainfall value to zero. At hour 50, update the RA card parameter yet again making it a negative value, denoting evaporation, and at hour 60 update the RA card again to set the value back to zero.

Compensating For The Rotation Of The Earth

The Coriolis force is the inertial force caused by the Earth's rotation. RMA2 provides two ways to control Coriolis calculations:

- Average Latitude Of The Mesh
- Specifying Latitude By Material Type



Warning: Instabilities on the southwest quadrant of the solution have been observed when using Coriolis Force correction in the Northern hemisphere. If this is a concern, a workaround is to apply graduated values of latitude (Coriolis). This is done by setting the latitude value for elements near the area of instability to zero, and increment the value of latitude in rows or layers of elements toward the point in the mesh where you want the value of latitude to be the true latitude. An increment of about 25% of the true latitude should be sufficient.

Average Latitude Of The Mesh

You can provide the *average* local latitude of the entire mesh in one step on the LA card. This is useful when you want to include one Coriolis force to be applied throughout the entire mesh domain, as opposed to Specifying Latitude By Material Type.

Specifying Latitude By Material Type

If the study domain is very large, you may want to provide several Latitude assignments which will encompass all of the modeled area. The LA card with the T option permits you to specify the latitude according to material type. This method will provide more customized Coriolis force calculations than when specifying globally the Average Latitude Of The Mesh. Alternatively, this material type option permits you to gradually decrease the Coriolis forces if instabilities on the southwest quadrant are observed.

Applying Wind Friction

RMA2 has several optional wind shear stress formulations. This gives you considerable flexibility in handling wind effects which are extremely difficult to implement in a two dimensional model. Wind is de-emphasized and generally not used in this program because wind-driven currents are three dimensional in nature. However, in systems that are generally shallow with strong wind influences, wind stress may be required to obtain proper setup and system circulation.



Note: Modeling experience with RMA2 running wind suggest that the wind speed and direction be gradually "ramped" over time between extreme values.

Applying wind friction is a two step process.

- Specify The Controlling Wind Formulation
- Specify Wind Speed And Direction

Once wind friction has been established, it remains in effect until a change is specified.



Tip: Remember that wind friction can be included or removed at any timestep. To remove wind friction effects after they have been activated, set IWIND=0 on the BWC card.

Specify The Controlling Wind Formulation

RMA2 currently provides eight formulations from which to choose when applying wind friction.

1. RMA2 Original Formula
2. Van Dorn Formula
3. Wu Formula
4. Safaie Formula
5. Ekman Formula
6. Generic Formula
7. Van Dorn Formula with defaults
8. Wu Formula with defaults

The wind stress formula is selected by setting the IWIND parameter on the BWC card. The majority of numerical modelers at WES have opted for the Wu formulation. See the BWC card for further information.

Specify Wind Speed And Direction

The wind speed and direction may be applied globally throughout the mesh domain, or by material type, element, or individual node on the BW card with the appropriate option.

Simulating With Storms

RMA2 provides the capability to include storm fronts in your simulation. This is accomplished by generating spatial variations in wind speed and direction for each time step based upon a user specified set of criterion for the storm(s). These criteria include factors such as:

- storm path
- orientation of the storm along the path
- storm speed
- shape
- temporal growth and decay constant for wind



Note: Storm rotation is based on the Coriolis forces for the northern hemisphere.

To date, the two dimensional Mississippi River Gulf Outlet study is the only application at WES in which storms were used to provide the wind stress. Because of lack of experience in this area, this feature remains experimental.

Storms In A Simulation

Many open water systems are strongly influenced by meteorological forcings which can apply pressures at the water surface. These wind stresses can be from a variety of storm types. The wind effects can generally be classified either as steady in nature or associated with a dynamic event. The ability of RMA2 to adequately simulate a real-world event must be carefully evaluated for each of these classes.

The response of a water body to wind forcing will generally be three-dimensional. The surface wind stress will drive a surface current in the direction of the wind. The influence of that surface wind drift will then be dependent on the water depth and the local shoreline geometry. If the currents are directed to a shoreline, the movement of surface water will result in a piling up of water on that shoreline. As the water level is raised on the shore, a hydrostatic pressure gradient is developed perpendicular to the shore, which will in turn drive a return flow away from the shore in the bottom portion of the water column. This vertical circulation pattern will moderate the increase in the water level at the shoreline. The strength of that return flow will be dependent on the water depth; generally stronger in shallower water.

The process is however a dynamic one. During the initial movement of water toward the shoreline, there is insignificant return flow at the bottom. Only after the water level is dramatically affected does the three-dimensional circulation develop. Wind events associated with stationary meteorological conditions may generate persistent unidirectional winds for prolonged periods of time, which may fully develop vertical circulation cells in the water column. However, there are a large number of wind events that are of a shorter duration and do not involve a fully developed vertical circulation response.

RMA2 Considerations

The three-dimensional nature of the circulation affects how appropriate the application of a depth-averaged model will be to the problem. It has been shown that for a steady-state type of wind forcing, RMA2 will overestimate the setup of the water levels at the shoreline. However, for dynamic events, the model will generally perform well for the initial movement of water. If the dynamics of the storm lead to a change in the wind stress such that a steady-state condition is never approached, then RMA2 will yield reasonable results.

If the shape of the shoreline is more complex, horizontal variations of the water surface may result in localized return flows that do not involve significant vertical circulation. Such circulation patterns are directly analogous to rip tides along a beach in response to the radiation stress of the waves.

The ability of RMA2 to properly simulate wind forcing hydrodynamic response is limited to cases where steady-state conditions are *not* approached. RMA2 will result in a raised water surface, or “setup”, at the shoreline, but because of the depth averaging, will have no return flow. Technically, RMA2 gives the appropriate “average velocity” for such conditions. However, if water quality or any dispersive processes are of interest, the results must be interpreted with care. In addition, the water surface setup will normally be overestimated.

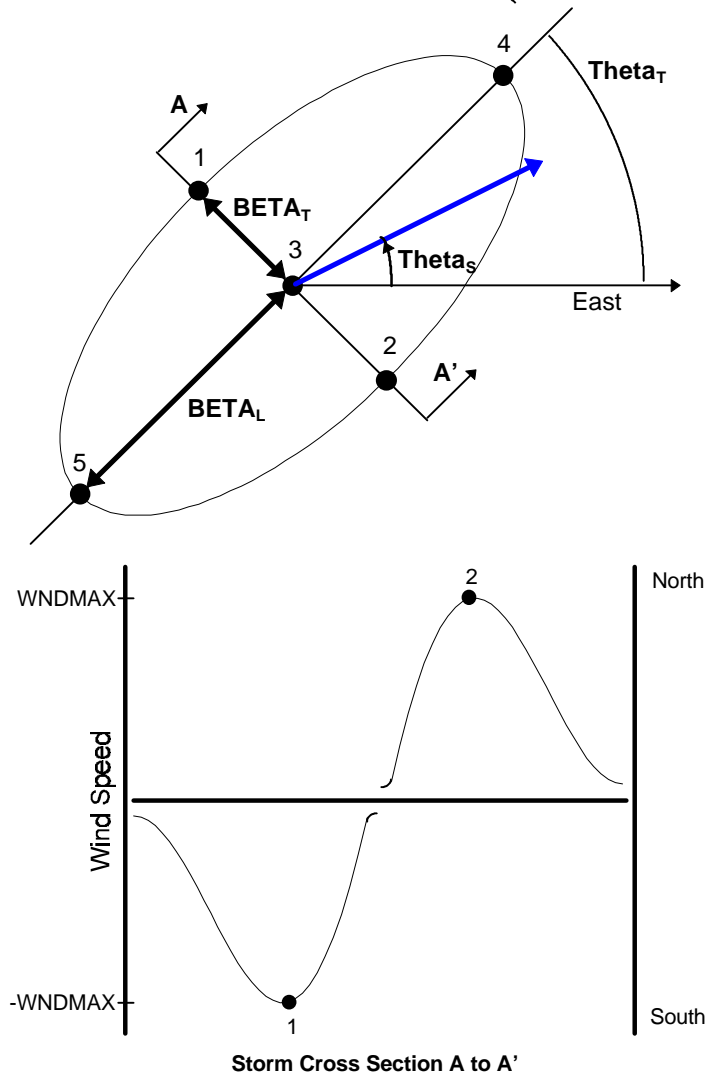
RMA2 can adequately handle certain types of dynamic events. The response of a system to a frontal passage or tropical storm can be evaluated with some confidence level by RMA2, provided the storm does not stall, which will lead to some steady-state issues.

Defining Storm Events

The representation of storms within RMA2 is provided as a means of easily developing relatively complex wind fields. A “storm” in RMA2 is defined as a specific tracking of the location of the storm center as a function of time, and the characteristics of that storm.

A storm is defined using the BWS card. Each BWS card creates a single storm. You may simulate multiple storms (presently dimensioned to two) to describe complex wind patterns by including additional BWS cards.

The storm characteristics are as described below. Refer to this illustration.



The Storm Reference Point

There are several options regarding the placement of the storm reference point relative to the numerical mesh. Referring to the figure, the reference point, *ISTYPE*, can be defined as either:

1. The location of maximum winds along the positive minor axis

2. The location of maximum winds along the negative minor axis
3. The center of the storm
4. The location of maximum winds along the positive major axis
5. The location of maximum winds along the negative major axis

These options are designed for convenience in obtaining the desired wind direction at a particular place and time when used in conjunction with other variables.

Time And Place Of The Occurrence Of Maximum Winds

The initial position of the storm is back tracked based on the storm tracking speed and direction of storm movement. The RMA2 simulation time, TREF, in decimal hours, is when the storm will arrive at the designated reference node, NREF, where NREF defines the location of the storm reference point. This places the peak winds relative to a specified time of the model simulation.

The pairing of location and time, combined with the storm path direction are used to develop the full storm track during the simulation. Based on the reference storm location and time specified, RMA2 will perform all calculations to determine past and future storm movements.

Storm Tracking Speed

The speed, SSPD, in miles per hour (mph) of the storm across the study domain.

Storm Major Axis Dimension

The longitudinal distance, $BETA_L$, from the storm center to the point of maximum wind speed, in feet or meters depending upon the setting of the SI card. The major axis is generally defined as the long axis.

By defining the major spatial axis as *large* relative to the minor axis dimension, a cold front storm can be simulated. This will normally require the major axis to be *very* large relative to even the model mesh dimensions. A Hurricane type storm can be simulated by choosing both the major and minor axis as having the same value, creating a circular storm pattern.

Storm Minor Axis Dimension

The transverse distance, $BETA_T$, perpendicular to the major axis from the storm center to the point of maximum wind speed, in feet or meters depending upon the setting of the SI card.

Maximum Wind Speed

The maximum wind speed, WNDMAX, in miles per hour (mph) will define the maximum winds along the ellipse defined by the major and minor axes.

Minimum Wind Speed

The minimum wind speed, WNDMIN, in miles per hour (mph) will define the minimum winds within the storm.

Growth And Decay Constant Relative To Maximum Winds

The exponential temporal decay coefficient, DECAY (hr), is a Gaussian variance of wind speed relative to the time of maximum winds. This parameter facilitates the

specification of daily wind patterns and tropical storm growth. This constant is normally derived from field data. A typical value for daily winds would be 6 hours.

Orientation Of The Major Axis Of The Storm

The direction THETA_T of storm orientation in degrees counter-clockwise relative to the positive x axis. This is significant for frontal passages to get the spatial variation of winds properly prescribed over the model mesh.

Direction Of Storm Movement

The direction toward which the storm is moving, THETA_S , is the storm track in degrees counter-clockwise from the positive x axis.



Note: This direction does not have to match the orientation of major axis.

Adjusting RMA2 Calculations

RMA2 allows you to modify the temporal derivative calculation. Typically, you will never change this. However, if you feel you must, the CA card is what you use to change the parameter.

The variable USERCA1 on the CA card is used to calculate the temporal derivative. Generally, values of USERCA1 which are closer to 1.0 will provide more model stability with less accuracy, while values closer to 1.6 provide more accuracy with less stability. A value of 1.5 yields a second order Taylor series expansion. With small time steps, the value of USERCA1 has little or no effect. With larger time steps, try values closer to 1.0 if model stability is a problem.

It has been mathematically derived that the default value of 1.6 will tend toward an unstable solution. However, based upon discussions with the primary author (Dr. Ian King of RMA), a large collection of experience has shown that for a Crank Nickolson parameter between 0.62 and 0.69, the value of 1.6 performs well and should remain as the default.

Run Control

Overview

RMA2 obtains its run control data from a set of input data cards provided in run control files. These data cards conform to what is known as HEC format. Each card has 80 columns of characters in which to hold its data. The first three columns are reserved for the card name, so there are actually 77 columns in which to hold functional data.

A data card occupies a single line in the run control input file. The card line is divided into data fields, of which the first contains the card name, and is designated as field 0. Field 1 begins the actual data for the card. RMA2 uses a free field format for data card input. Each item of data constitutes a field. There can be as many fields on a card as there is room within the 80 columns of characters.



Note: If more than one card modifies a variable, the last card rules.

Summary Of Run Control Data Cards

Although most of the data cards are independent of order, there are indeed some dependencies.

The following table lists all the data cards available in RMA2, and whether or not the card is required. The order of the list is the suggested order in which the cards should appear in the RMA2 run control file.

Card	Content	Revisable?	Required?
T1-T2	Title cards	No	No
T3	Title card	No	Yes
CO	Comments (anywhere except first line)	---	No
\$M	Machine type identifier	No	Yes
\$L	Input/output file numbers	No	No
SI	System international units	No	No
G1	Geometry, general geometry parameters	No	Replaced
GC	Geometry, continuity check lines	No	No
GCL	Geometry, continuity check lines	No	No
GE	Geometry, element connection table	No	No

GN	Geometry, nodal coordinates, bottom elevations	No	No
GS	Geometry, scale factors	No	No
GT	Geometry, element type (IMAT)	Yes	No
GV	Geometry, eddy viscosity tensor	Yes	No
GW	Geometry, one dimensional cross sectional properties	No	No
GZ	Geometry, bottom elevations	No	No
CA	Special calculation variables	Yes	No
DE	Wet/dry by element	Yes	No
DM	Wet/dry by marsh porosity	No	No
EV	Turbulent exchange and Manning's n coefficients	Yes	Yes, If EX and EY and HN not present
EX	Turbulent exchange coefficients, e, x-plane	Yes	No
EY	Turbulent exchange coefficients, e, y-plane	Yes	No
FC	Flow control structures	Yes	No
FD	Fluid density	Yes	No
FT	Fluid temperature	Yes	No
HN	Hydraulics, Manning n-values	Yes	No
IC	Initial conditions, water-surface elevation	No	Yes
LA	Latitude	Yes	No
PE	Peclet control of eddy viscosity	Yes	No
RA	Rainfall and evaporation	Yes	No
RD	Roughness assignment by depth	Yes	No
TI	Timing, number of iterations counter	Yes	Yes
TO	Timing for binary output write	No	No
TR	Trace printout controls	Yes	Yes
TRN	Trace printout node list	No	No
TS	Timing for writing to (u,v,h) solution	Yes	No
TV	Timing of vorticity iteration	Yes	No
TZ	Timing, of simulation	Yes	Yes
BCC	Boundary condition control parameters	Yes	No
BA	Boundary, azimuth of flow	Yes	*
BS	Boundary, current speed	Yes	*
BH	Boundary, water-surface elevation	Yes	*
BQ	Boundary, unit discharge	Yes	*
BRA	Boundary, reflection/absorption	Yes	*
BRC	Boundary, rating curve	Yes	*
BWC	Boundary, wind formulation control	Yes	No
BW	Boundary, surface IWND data field	Yes	No
BWS	Boundary, wind storm	Yes	No
BCN	Boundary condition specified by node	Yes	*
REV	Revise the coefficients or boundary conditions in mid-time step	---	No
END	End of boundary condition specifications for the	---	Yes

	time step		
STO	Stop the simulation	---	Yes

* *At least one of these specifications of boundary conditions is required.*

Input Variables

The following is a table of input variables which are the parameters found on the RMA2 data cards (except those in *italics*, which are not actually input on the card, but are related to the card). The table lists the variable name, a description, and the card or cards to which it is associated.

Variable	Description	Content
AC1	Distance from the average regional bed elevation to the minimum regional bed elevation	DM
AC2	Transition range of the distribution	DM
AC3	Minimum wetted surface area factor	DM
AC4	Minimum regional bed elevation	DM
AC1X	Constant for rating curve	BRA, BRC
AC2X	Multiplier for rating curve	BRC
AC3X	Base elevation for rating curve	BRC
AC4X	Exponent for rating curve	BRC
ALFAK	Angle at a node, in radians	BA
AO	The bottom elevation of each node	GN, GZ
ASEC	Coefficient for the vorticity equation	VO
BETA	Reflection/absorption boundary coefficient	BRA
BETAL	Standard deviation of spatial distribution function in direction of storm's movement	BWS
BETAT	Standard deviation of spatial distribution function in direction of transverse to storm path	BWS
CORD	The (x,y) coordinates of the node	GN
DECAY	Exponential temporal decay for wind speed (hr^{-1})	BWS
DELT	Length of computation time step	TZ
DSEC	Coefficient for the vorticity equation	VO
DSET	The water depth at which a wet node is considered to become dry	DE
DSETD	The water depth at which a dry node becomes rewet	DE
ELEV	The average initial water-surface elevation over the mesh	IC
EPSXX	Scaling factor for computed eddy viscosity in xx direction	PE
EPSXY	Scaling factor for computed eddy viscosity in xy direction	PE
EPSYX	Scaling factor for computed eddy viscosity in yx direction	PE
EPSYY	Scaling factor for computed eddy viscosity in yy direction	PE
FLD	Any alphanumeric user comment	CO, END, REV, STO
FLZ3	Flow control equation, base flow	FC
FLZ4	Flow control equation, relational coefficient	FC
FLZ5	Flow control equation, reference elevation or head difference	FC
FLZ6	Flow control equation, exponent	FC
FLZ7	Flow control equation, direction of flow (radians counterclockwise from + x-axis)	FC

GPEC	Peclet number	PE
HFX	Head specification along a continuity line	BH
HMIN	Initial depth for one dimensional elements	IC
IBHO	Scheme for saving RMA2 binary (u, v, h) solution	TS
IBUP	Logical unit number for dynamic boundary conditions input to RMA2	\$L
IBVO	Scheme for saving RMA2 binary vorticity solution	TS
IC1	Card group identifier, all cards	ALL
IC3	Data type identifier, some cards	SOME
ICON	Continuity line number	BRA
IVRSID	Computer identifier	\$M
<i>IDEN</i>	Counter for the number of node that have a non-default fluid density assignment (self-count)	FD
<i>IDNOPT</i>	Marsh porosity option on/off switch	DM
IECHO	Switch to control echo printing of coded input data records	TR
IFILE	Logical unit number for reading GFGEN binary geometry	\$L
IFINO	Logical unit number for RMA2 to write results for transfer to RMA4 or SED2D	\$L
IGEON	Logical unit number for GFGEN geometric data file	\$L
<i>IHGEN</i>	Dynamic counter for input of number of lines across which elevation will be specified (self-count)	BH
IHOTN	Logical unit number for file containing initial conditions	\$L
IHOTO	Logical unit number for RMA2 to write-restart file	\$L
IMAT	The element type (n-value and eddy coefficients)	GE, GT, LA, PE
IOUT	Logical unit number for writing full results listing	\$L
IPEC	On/off switch to control eddy viscosity by Peclet number	PE
IPRT	Switch to print element input data, initial conditions and n-values	TR
<i>IQGEN</i>	Dynamic counter for input of number of lines across which total flow will be specified (self-count)	BQ
IRUFF	On/off switch for auto Manning's n-value	RD
ISPRT	Logical unit number for writing summary results listing	\$L, TR
ISTART	Starting number for global assignments (local variable)	LA, RD
<i>ISTGEN</i>	Dynamic counter for input of number of lines across which an elevation-flow relationship will be specified	BR
ISTYPE	Storm reference point	BWS
ITRACE	Trace subroutine calls and controllers (debug)	TR
ITSI	Number of time steps between successive full results listings	TR
IVOR	On/off switch for vorticity calculations	VO
IWIND	Control for wind field input	BCC, BWC
IWMX	Wind speed unit flag	BWC
IWR33	Logical unit number for writing detailed steady state parameters when using automatic features (viscosity and roughness)	\$L

<i>JCH</i>	Continuity line number for head boundary (J = JCH)	BH
<i>JCQ</i>	Continuity line number for total flow (J = JCQ)	BQ
JCR	Continuity line number for rating curve	BRC
LI	The number of iterations between checks for dry nodes	DE
LINE	Corner node numbers for continuity check	GC, GCL
LMT	Total number of corner nodes on a given continuity line	GC
MBAND	Restart in mid-iteration (flag)	TZ
METRIC	System International (flag)	SI
NBX	Number of nodes with boundary conditions specified	BCC, G1
<i>NCFLW</i>	Number of flow control structures (self-count)	FC
NCYC	Number of time steps simulated	TZ
NFIX	Array containing logic flags for boundary condition	BCN
<i>NFIXH</i>	Reordering list read from GFGEN	GFGEN GO
NGOODMAX	Number of good passes through a series of (u, v, h) and vorticity calculations before the solution has converged	TV
NITI	Number of iterations for initial solution (or steady state computation)	TI
NITN	Number of iterations for each dynamic computation	TI
NJN	Flow control structure identifier (IMAT \geq 904)	FC
NJT	Flow controller type	FC
<i>NMAT</i>	The number of different sets of turbulent exchange coefficients and Chezy and/or Manning coefficients	EV, EX, EY
NOP	Nodal point-element connection table for RMA2	GE
NOPTV	Logical unit number for writing vorticity solutions in RMA4 format	\$L
NPASS1	Maximum number of passes between the steady state (u, v, h) and vorticity iterations	TV
NPASS2	Maximum number of passes between the dynamic (u, v, h) and vorticity iterations	TV
NREF	Nodal point reference for storms	BWS
<i>NSID</i>	Dynamic counter for number of elements for which element inflow is desired	BQ
NSPLPT	Array containing nodes for summary results listing	TRN
NSTART	Starting time-step number used to skip through boundary condition data for restart	TZ
NVITI	Maximum number of steady state vorticity iterations	TV
NVITN	Maximum number of dynamic vorticity iterations	TV
OMEGA	Latitude of mesh (approximate average)	G1, LAT
ORT	Eddy diffusion and n-value array by element type	EX, EY, HN, EV
QDIR	Flow direction along continuity line	BQ
QF	Total flow along a continuity line	BQ
QXP	Factor for distribution of flow by depth	BQ
QVEC	The unit discharge (cfs/ft)	BQ
RCMIN	Minimum radius of curvature with vorticity	VO
RDR0	Maximum Manning's n for non-vegetated water	RD

RDD0	Depth at which vegetation effects roughness	RD
RDCOEF	Roughness by depth coefficient	RD
RON	The array of nodal fluid density	FD
SIDF	Element inflow (per unit area)	BQ
SPEC	Array containing boundary condition specifications	BCN, BH
SS1	Left side slope for one dimensional nodes	GN, GW
SS2	Right side slope for one dimensional nodes	GN, GW
SSDCRT	Steady state satisfactory depth (convergence) criterion	TI
SSPD	Storm speed, mph	BWS
SSVCRT	Steady state vorticity convergence criterion	VO
SSWSE	Steady state water surface elevation	BRA
TAREA	Surface area of tidal storage beyond the boundary	BRA
TAX	Wind direction (degrees counterclockwise from + x-axis)	BW
TBINRY	Array containing user selected hours to save to binary final results file	TO
TEMP	Average initial water temperature	FT
TH	Azimuth of x-direction of an element for specifying eddy diffusion coefficients	GE, GV
THETA	Direction of flow (radians measured counterclockwise)	BRA, BRC
THETAS	Direction of storm track	BWS
THETAT	Orientation of storm	BWS
TITLE	Character identifier for the run and all output files	T1-T3
TMAX	Total time in decimal hours	TZ
TREF	RMA2 simulation time of storm arrival at 'NREF'	BWS
TWX	Wind speed	BW
UNOM	Initial velocity for one dimensional elements	IC
USDCRT	Dynamic satisfactory depth criterion	TI
USDVCC	Dynamic vorticity convergence criterion	VO
USERCA1	User selected variable for temporal derivative calculation	CA
VOR	Vorticity value	BV
VPEC	Initial guess when using automatic Peclet number	PE
VVEC#	Current speed (fps) at a node	BS
WC1	Wind control coefficient	BWC
WC2	Wind control coefficient	BWC
WC3	Wind control coefficient	BWC
WC4	Wind control coefficient	BWC
WDMC1	See AC1	DM
WDMC2	See AC2	DM
WDMC3	See AC3	DM
WDMC4	See AC4	DM
WIDS	Storage with associated one dimensional node	GN, GW
WIDTH	Channel width for nodes	GN, GW
WNDMAX	Maximum wind speed (mph)	BWS
WNDMIN	Minimum wind speed (mph)	BWS

XSCALE	Scale factor for x-coordinates	G1, GS
YSCALE	Scale factor for y-coordinates	G1, GS
ZSCALE	Scale factor for z-coordinates	GS

RMA2 Execution Job Sheet

JOB EXECUTED _____ DATE OF RUN ____/____/____ TIME OF RUN _____

JOB PRINTED _____ SUBMITTED BY _____ CPUs ____ PRIORITY_____

PURPOSE:

SIMULATION TIME: Start _____ Finish _____

FILES:

Primary Run Control File (.rc2)	_____
Geometry file from GFGEN	_____
RMA2 Hotstart input	_____
RMA2 alternate BC file	_____
RMA2 Full results listing	_____
RMA2 summary results listing	_____
RMA2 Hotstart output	_____
RMA2 final results file	_____
RMA2 vorticity results file	_____
RMA2 info on EV/n-value	_____

GRAPHICAL ANALYSIS:

VELOCITY vector plots	_____
Time Series plots	_____

RMA2 Data Cards

This section describes all of the data cards used by RMA2. Every effort has been made to describe each card in a clear and complete manner. However, the appropriate sections in the manual text should be consulted when further explanation is desired.

\$L Card: Input/Output File Control

Optional

Card Description: Used to specify what types of files RMA2 will read and write.

Field	Variable	Value	Description
0, C 1-2	IC1	\$L	Card group identifier.
1	IHOTN		Hotstart input.
		0	Initial conditions for RMA2 will be coded in the run control file containing this \$L card.
		+	Initial conditions will be read from logical unit #63 (The Hotstart results from the previous run).
2	IHOTO		Hotstart output.
		0	No Hotstart file will be written.
		+	RMA2 will write a Hotstart file on unit #62.
3	IGEON		GFGEN geometric data, input for RMA2.
		0	All geometry will be coded in the RMA2 data set.
		+	GFGEN's geometric data opened on unit #60.
4	IFINO		RMA2 solution output (binary final results of hydraulic calculations).
		0	No RMA2 solution output is saved.
		+	The logical unit number for RMA2 solution output.
5	IBUP		Alternate dynamic boundary conditions (input).
		0	No alternate boundary conditions file.
		+	Alternate boundary file read from unit #61.
6	IOUT		Full (standard) results listing file.
		0	No full listing will be created.
		+	Full results listing will be created on unit #3.
7	ISPRT		Summary results listing by node option.
		0	No special list of nodes written.
		+	Summary results listing created on logical unit #59 (TRN card(s) required).
8	IWR33		Detailed steady state parameters when using automatic features (RD card and/or PE card). <i>Version 4.28 or higher.</i>
		0	Off
		+	Listing created on logical unit #33.
9	NOPTV		Save vorticity calculation to a file having an RMA4 binary solution format. <i>Version 4.30 or higher.</i>
		0	Off
		+	Vorticity calculations saved on logical unit #98.



Note: Any default logical unit number can be overruled by coding a negative number in the data field for that unit number. The logical unit number will then be the absolute value of the negative number specified.




Note: If no \$L card is present, all file options are off except the full results listing file.

\$M Card: Machine Identifier

Required

Card Description: Used to specify the type of computer on which RMA2 will be running. The machine type determines the word size used for the matrix buffer temporary files.

Field	Variable	Value	Description
0, C 1-2	IC1	\$M	Card group identifier
1	IVRSID		Controller for record length and word size for front solver buffering. Choose the value from one of the following based on the type of computer system on which the model will be running. 1 Intel x86 and Pentium Microprocessors (PC). Direct access record length is unlimited and is defined in terms of bytes. 2 Prime Mini-Computer. Direct access record length is unlimited and is defined in terms of small words (i.e. 2 bytes). 3 DEC VAX. Direct access record length, limited to 32K bytes and defined in terms of long words (4 bytes). 4 HP or ALPHA workstations, Apple MAC II using ABSOFT FORTRAN, Definicon 020 Board, or DEC VAX to avoid short record limit. Direct access defined using multiple sequential access files that are opened as required.  Note: Many files may be left on disk. 5 Cray or Cyber-205. Direct access defined for systems using 64 bit or 8 byte words and where record lengths are defined in bytes. 6 Same as option 4 above, except the names of the files that are opened will not contain a '.' (dot). 8 Same as option 4 above, except PAUSE statements in the program are processed for interactive sessions. (Recommended for Apple MAC)



Note: If no \$M card is supplied, IVRSID = 4 by default.



Example:

```
.  
.  
$M      1      Specifies DOS PC  
.
```



See also: "What Kind Of Computer Do You Have?" on page 23.

BA Card: Boundary, Azimuth of Flow

Version 4.20 or higher.

Optional

Card Description: Used to specify the azimuth (direction of flow) in radians at the specified nodes. The BA card is typically followed by a BS card describing the same nodes. The BA card specifies the direction and the BS card specifies the speed.

Field	Variable	Value	Description
0, C 1-2	IC1	BA	Card group identifier.
0, C 3	IC3	N	Card type identifier.
1	J1	+	The boundary node number for azimuth ALFAK(J1).
2	ALFAK(J1)	+	The angle at node J1, in radians counterclockwise from the <i>x</i> -axis, of the velocity vector (or unit discharge vector).
3	J2	+	Another boundary node number (if desired).
4	ALFAK(J2)	+	The angle at node J2.
5-10	etc.	+	You may continue coding node and angles until the card is full. If necessary continue coding with another BA card.



Note: Either BCC cards or comparable data on BA through BRC cards is required.



FastTABS Note: As of this writing, FastTABS does not support BA cards.




See also: “Changing The Direction Of Flow” on page 73.

BCC Card: Boundary Condition Control Parameters

Version 4.20 or higher.

Optional

Card Description: Used if you wish to revise a boundary condition and update parameters between dynamic time steps.

Field	Variable	Value	Description
0, C 1-2	IC1	BC	Card group identifier.
0, C 3	IC3	C	Card type identifier.
1	DELT	+	The delta time step length in decimal hours (DELT is revised only if the value is greater than zero).
2	NBX	0, +	The number of BCN cards for which boundary conditions are specified. If zero, the program will <i>self-count</i> the number of BCN cards for this time step (if and only if an END card is used to mark the end of the time step).
3	IWIND	-, 0, +	Control for wind field input.  Note: Reference the BWC card for IWIND values.



Note: If this card is present, any boundary condition parameters specified on previously read cards (TZ card, BW card, or previous BCC cards) will be overruled.



See Also: “Revising Boundary Conditions During A Simulation” on page 73, TZ card, REV card

BCN Card: Boundary Conditions by Node

Version 4.20 or higher.



Optional

Card Description: Assigns a boundary condition to the specified node.

The three primary boundary condition parameters can be coded on this card type: x velocity component, y velocity component, and water surface elevation (head). A five-digit number that tells the program the parameter type is coded, as well as the nodal point number. Code one BCN set for each boundary node.



Note: Both corner and mid side boundary nodes require boundary conditions.

Field	Variable	Value	Description
0, C 1-2	IC1	BC	Card group identifier.
0, C 3	IC3	N	Card type identifier.
1	J	+	Code the node number receiving the boundary condition.
2	NFIX(J)		This 5 digit number tells RMA2 what type of boundary condition to use. Coding leading zeros is not required.
		00200	Water surface elevation (head) specified at node J in data field 5.
		01000	Assign a slip flow boundary at node J.  Note: This is not required in Version 4.2 or higher because slip flow boundaries are automatically generated for edges.
		01200	Combination slip flow with head specified at node J. Supply the water surface elevation (head) value in data field 5.
		11200	Assign a combination of x - and y -velocity with a water surface elevation (head). Supply values in data fields 3, 4, and 5.
			 Warning: Beware of over-specification.
		11000	Both x - and y -velocity components are specified at node J. Supply these values in data fields 3 and 4.
3	SPEC(J,1)	-, 0, +	The x -component of velocity, ft/sec or m/sec.
4	SPEC(J,2)	-, 0, +	The y -component of velocity, ft/sec or m/sec.
5	SPEC(J,3)	-, 0, +	The water surface elevation (head), feet or meters.



Note: If BCN cards are not used, comparable data on BA cards through BRC cards is required. Code one node per BCN card.

If NBX on the BCC card is non-zero, and an old format style alternate dynamic boundary file is specified, then BCN cards should be the last set of input cards for a given time step.



See also: “Specifying Boundary Conditions” on page 29.


BH Card: Boundary Head

Version 4.20 or higher.

Optional

Card Description: Used to assign a water surface elevation at the specified node(s) as an alternate to BCC cards.

NFIX (see BCN card) is assigned xx200 at each node where boundary head data exist, where the xx's denote optional values which may be assigned by a BS card or BQ card.

Field	Variable	Value	Description
0, C 1-2	IC1	BH	Card group identifier.
0, C 3	IC3		Card type identifier.
		b (blank)	Specifies Option 1: The water surface elevation will be used for all boundary nodes equal to or greater than J.
			 Warning: Beware of over-specification.
		N	Specifies Option 2: The water surface boundary condition is coded at node J.
		L	Specifies Option 3: The water surface boundary condition is coded by continuity check line number J.
1	J		The node or continuity check line number as specified by the value of IC3 above in data field 0, column 3.
		+	Option 1: The <i>starting node</i> number.
		+	Option 2: The <i>node</i> number.
		+	Option 3: The <i>continuity check</i> line number.
2			Code SPEC(J) or HFX(J) depending on which option was chosen above with IC3 in data field 0, column 3.
	SPEC(J)	-, 0, +	Option 1: Water surface elevation (ft or m) for all nodes equal to or greater than J.
	SPEC(J)	-, 0, +	Option 2: Water surface elevation (ft or m) for node J.
	HFX(J)	0, +	Option 3: Water surface elevation (ft or m) for the continuity check line J.



Note: Only one value per BH card is permitted (comments may follow if you wish).



See also: “Specifying Boundary Conditions” on page 29.


BQ Card: Boundary Discharge


Version 4.20 or higher.


Optional


Card Description: Used to assign a discharge at the specified node(s), element(s), or line(s).

This card type can be used instead of the BS cards. The program will assign NFIX on the BCN card as 13x00 or 31x00 where the *x* denotes the values to be assigned by BH card data.

Field	Variable	Value	Description
0, C 1-2	IC1	BQ	Card group identifier.
0, C 3	IC3		Card type identifier.
		b (blank)	Specifies Option 1: Use the unit discharge in data fields 2 and 3 for all boundary nodes equal to or greater than J.
		N	Specifies Option 2: The <i>node</i> number for inflow value.
		E	Specifies Option 3: The <i>element</i> number for element side inflow.
		L	Specifies Option 4: The <i>continuity check line</i> number for inflow value.
1	J		The node, element, or continuity check line number as specified by the value of IC3 above in data field 0, column 3.
		+	Option 1: The <i>starting node</i> number.
		+	Option 2: The <i>node</i> number.
		+	Option 3: The <i>element</i> number.
			Option 4: The <i>continuity check line</i> number.
2	QVEC	-, 0, +	Option 1: The unit discharge, (ft ³ /s/ft, or m ³ /s/m), at all nodes equal to or greater than J. Option 2: The unit discharge (ft ³ /s/ft, or m ³ /s/m) at node J.  Note: The program will calculate the <i>x</i> and <i>y</i> components of unit discharge vectors from QVEC by using the azimuth given on the BA card. The signs of the <i>x</i> and <i>y</i> unit discharge components are calculated from azimuth and grid orientation specified on the BA card.
	SIDF	-, 0, +	Option 3: The elemental <i>inflow</i> per unit area (for 2D) or unit length (for 1D) as appropriate for the element. Positive values represent inflow, negative values represent outflow.
	QF	0, +	Option 4: The total flow (ft ³ /s, or m ³ /s) crossing the continuity line.

3	QDIR	-, 0, +	<p>Option 4: The direction of flow in radians measured counterclockwise from x-axis.</p> <p> Note: The program adjusts the boundary directions to maintain flow parallel to the edges for nodes at the ends of the continuity line.</p>
4	QXP	0 to 1	<p>Option 4: Distribute flow by depth. (default = 0) <i>Version 4.296 or later.</i></p> <p>Decimal values between 0 and 1 are allowed. If QXP = 0, the total flow is equally distributed among all nodes across the continuity check line. If QXP = 1, the total flow is weighted by the depth of each node occurring on the line (shallow nodes get less flow, deeper nodes get more flow).</p>

 **Note :** Only one set of data per BQ card is permitted (comments may follow if you wish). If a BQ card is not used, then comparable data must be provided on either BCC cards or on BA cards through BRC cards.

 **Tip:** If specifying discharge along a continuity line (GC card), the continuity line is recommended to extend from bankline to bankline.



 **See also:** “Specifying Boundary Conditions” on page 29, “Element Inflow And Outflow” on page 83.


BRA Card: Boundary, Reflection/Absorption

Version 4.26 or higher.

Optional

Card Description: A numerical means of controlling the level of reflected energy at the boundary, from full reflection to full absorption (radiation boundary condition).

Field	Variable	Value	Description
0, C 1-2	IC1	BR	Card group identifier.
0, C 3	IC3	A	Card type identifier.
1	ICON	+	Continuity line number.
2	AC1X	-, 0, +	Average (net) river discharge ft ³ /sec or m ³ /sec , (Variable Q _o in the equation below).
3	BETA	0 to 1	Reflection/Absorption coefficient used to calculate A ₂ in the rating curve equation below. BETA = 1 specifies <i>total reflection</i> at the boundary. BETA = 0 specifies <i>total absorption</i> at boundary.
4	SSWSE	-, 0, +	The steady state solution for the water surface elevation at the first node on the specified continuity line number. If SSWSE is less than or equal to zero, RMA2 will incorporate the steady state solution (E _o) before advancing to dynamic.  Note: If Hotstarting a <i>steady state</i> run, use a value of -1. If Hotstarting a <i>non-steady state</i> (dynamic) run, you must supply SSWSE with at least 4 significant figures past the decimal.
5	THETA	-, 0, +	Direction of flow into the mesh in radians measured counterclockwise from the x axis.  Note: The program adjusts the boundary directions to maintain flow parallel to the edges for nodes at the ends of the continuity line.
6	TAREA	0, +	Surface area of tidal storage beyond the boundary. Used in defining standing wave flows. TAREA is the maximum tidal discharge divided by the maximum fall rate of water surface for a given time step. $TAREA = Velocity_{max} * Width * Depth / R_{max}$ $R_{max} = \Delta WSELV / (\Delta T * 3600)$

 **Note:** The special reflecting or non-reflecting (absorption) rating curve described by the following equation supplies a discharge to RMA2.

$$Q = Q_o + A_2 * (ELEV - E_o)^1$$

E_o in this equation will be supplied after the steady state solution has been computed. Best results are obtained if this feature is applied to a continuity check line of a one dimensional element.

 **See Also:** "Boundary Permeability (Reflection/Absorption)" on page 74.

BRC Card: Boundary Rating Curve


Version 4.20 or higher.

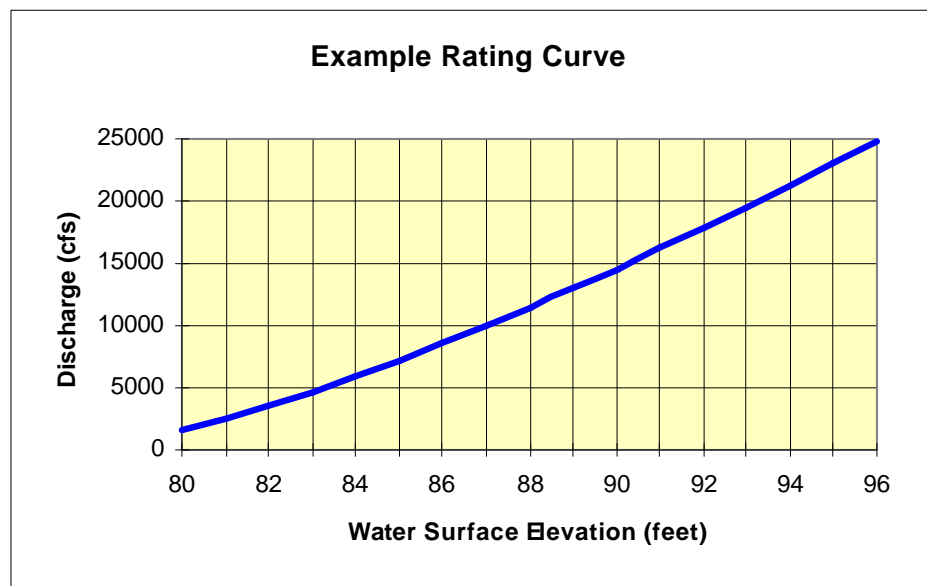
Optional

Card Description: Used to specify a boundary rating curve. The rating curve supplies a discharge to RMA2 using the following equation:

$$Q = A_1 + A_2 * (ELEV - E_0)^C$$

Field	Variable	Value	Description
0, C 1-2	IC1	BR	Card group identifier.
0, C 3	IC3	C	Card type identifier.
1	JCR	+	Continuity line number.
2	AC1X	-, 0, +	A ₁ in the above equation.
3	AC2X	-, 0, +	A ₂ in the above equation.
4	AC3X	-, 0, +	E ₀ in the above equation.
5	AC4X	-, 0, +	C in the above equation.
6	THETA	-, 0, +	Direction of flow (radians measured counterclockwise from + x-axis).

 **Note:** The boundary directions are adjusted to maintain parallel flow on the exterior edge.



$$Discharge = 1000 + 600 * (Elevation - 79)^{1.3}$$



See also: “Varying Discharge During A Simulation (Rating Curve)” on page 74.

BS Card: Boundary Current Speed

Version 4.20 or higher.

Optional

Card Description: Used to specify the magnitude of velocity for the specified nodes. The BS card must be accompanied by a preceding BA card describing the same nodes. The BS card specifies speed and the BA card specifies direction.

The magnitude of the velocity vector is coded on this card type. The input data program will convert BS card data to U and V velocity components using the azimuth on the preceding BA card. The sign of the component is calculated from its azimuth and the specified grid orientation. NFIX on the BCN card is assigned a value of 11x00 at each node having a BS value, where the *x* denotes a value to be assigned by the presence of BH card data.

Field	Variable	Value	Description
0, C 1-2	IC1	BS	Card group identifier.
0, C 3	ISI	N	Card type identifier.
1	J1	+	Node number.
2	VVEC1	0, +	Water current speed in ft/sec or m/sec at node J1. The sign will be determined from the azimuth of the vector.
3	J2	+	Next node.
4	VVEC2	0, +	Water current speed at node J2, ft/sec or m/sec.
5-10	<i>like 3 and 4 above</i>		Continue coding node number and speed sets until the card is full, then use another BS card.



Note: If BS cards are not used, then comparable data on either BCC cards or on BA cards through BRC cards is required.



FastTABS Note: As of this writing, FastTABS does not support BS cards.



See also: “Changing The Speed Of The Current” on page 74.

BV Card: Boundary Inflow Vorticity

Version 4.30 or higher.

Optional

Card Description:

Field	Variable	Value	Description
0, C 1-2	IC1	BV	Card group identifier.
0, C 3	IC3	L	Card type identifier.
1	J1	+	The continuity line number for inflow value.
2	VOR	0, +	The value of vorticity at the specified continuity line. Usually vorticity is set to zero at the inlet. All other vorticity boundary conditions are zero gradient and require no action to apply.
3	FLD		Comments



Note: The RMA2 vorticity boundary conditions may only be applied to one, and only one, boundary.




See also: “Bendway Correction (Vorticity)” on page 74, VO card.

BW Card: Boundary, Wind Speed and Direction

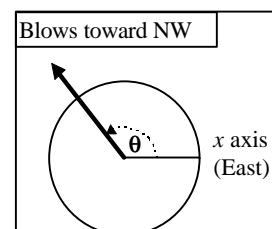
Version 4.20 or higher.

Optional

Card Description: Used to assign wind speed and direction.

Field	Variable	Value	Description
0, C 1-2	IC1	BW	Card group identifier.
0, C 3	IC3		Card type identifier.
		b	Specifies Option 1: The wind data in data field 2 and 3 of this card will be used for all nodes equal to or greater than J.
		N	Specifies Option 2: Wind data are coded by specific <i>node</i> number.
		E	Specifies Option 3: Wind data are coded by <i>element</i> number.
		T	Specifies Option 4: Wind data are coded by <i>material type</i> number (IMAT).
1	J		Code the node number, element number or material type as specified by IC3 above in data field 0, column 3.
		+	Option 1: The <i>starting node</i> number. Option 2: The <i>node</i> number. Option 3: The <i>element</i> number. Option 4: The element <i>material type</i> number.
2	TWX(J)	0, +	The wind velocity.  Note: Units are defined by variable IWMX on BWC card.
3	TAX(J)	-, 0, +	Direction toward which the wind is blowing, measured in degrees counterclockwise from the positive <i>x</i> -axis.

For Example, a SE wind, as reported by the conventional meteorological terms, blows toward the NW. This would be an angle of 135 degrees counterclockwise from a positive *x*-axis (with the positive *x*-axis oriented to the east).



Note: Generally, wind forces should not be necessary unless you have a large fetch length (possibly > 100 miles) in the area of interest. The required order for wind assignments should be BWC, BW *blank*, BWT, BWE, and BWN.




See also: “Applying Wind Friction” on page 85, BWC card.

BWC Card: Boundary Wind Formulation Control

Version 4.20 or higher.

Optional

Card Description: Specifies the numerical formula to be used when calculating wind stress forces. The BWC card must be followed by a BW card type in order to assign wind speed and direction.

Field	Variable	Value	Description
0, C 1-2	IC1	BW	Card group identifier.
1	IWIND		Control for wind field input.
		0	No wind input.
			Code the type of wind stress formula to use:
		1	Specifies Option 1: Original RMA2 formulation with <i>default</i> coefficients.
		2	Specifies Option 2: Van Dorn Formula with <i>user supplied</i> coefficients.
		3	Specifies Option 3: Wu Formula with <i>user supplied</i> coefficients.
		4	Specifies Option 4: Safaie Formula with <i>user supplied</i> coefficients.
		5	Specifies Option 5: Ekman Formula with <i>default</i> coefficients.
		6	Specifies Option 6: Generic Formula with <i>user supplied</i> coefficients.
		7	Specifies Option 7: Van Dorn Formula with <i>default</i> coefficients.
		8	Specifies Option 8: Wu Formula with <i>default</i> coefficients.
2	IWMX		Flag to identify the units of the wind parameters (recorded at anemometer height = 10 m).
		0	for miles/hour
		1	for meters/second
3	WC1		Specify data field 3-6 as needed Data field 3-6 will be the coefficients required for the wind formulation specified by the variable IWIND above.
			 See Wind Table 1 in "Wind Special Instructions" on page 117 for descriptions.
4	WC2		
5	WC3		
6	WC4		



Note: Except for option 1, all wind input will be specified in Metric units to conform with the units of the coefficient values used in the wind formulas, regardless of the setting of the SI card. The wind calculation is performed using Metric units. The results of the calculation are converted to match the units as specified by the SI card.

Wind speed and direction are specified on the BW card.

Unless noted otherwise, there are no default values for any parameters for formula options 2, 3, 4, and 6 as listed in Wind Table 1. If there is no value specified for a parameter, zero will be assumed.

If the chosen formula option is 1, 5, 7, or 8 then WC1, WC2, WC3, and WC4 are ignored and the program will assign default values.



Warning: Presently this method is in an evaluation stage and should be used with caution.



See also: “Applying Wind Friction” on page 85, BW card.

Wind Special Instructions

There are several options for including wind stress in the flow solution. The formulas available in RMA2 are taken from wind stress literature and are listed in Wind Table 2 on the following page. The wind stress formula is selected by setting the IWIND parameter on the BWC card. Wind Table 1 below provides instructions for coding the wind stress parameters.

Wind Table 1. Wind Special Instructions.

Option	Formula	WC1	WC2	WC3	WC4
2	Van Dorn	Currently unused. Code any value to skip this field.	Smooth water wind stress coefficient	Rough water wind stress coefficient.	Critical wind velocity for wave formulation (m/sec).
3	Wu	Currently unused. Code any value to skip this data field.	Wind stress coefficient.	Air Density (g/cm^3). <i>WC3 > 0, specify the air density in units of g/cm^3. WC3 £ 0, default of .001226 g/cm^3 is used (Dry air at 1 atm pressure and 15 deg C).</i>	Unused.
4	Safaie	Charnock's constant. <i>WC1 > 0, specify the constant. WC1 £ 0, default of .0332 is used.</i>	Dynamic roughness (cm).	Acceleration due to gravity (cm/sec^2) (affects wind formula only). <i>WC3 > 0, specify the acceleration in units of cm/sec^2. WC3 £ 0, default value of 979.965 cm/sec^2 is used.</i>	Air density (g/cm^3). <i>WC4 > 0, specify the air density in units of g/cm^3. WC4 £ 0, default value of .001226 g/cm^3 is used (Dry air at 1 atm pressure and 15 deg C).</i>
6	Generic	Currently unused. Code any value to skip this field.	Wind stress coefficient.	Empirically derived exponent.	Air density (g/cm^3). <i>WC4 > 0, specify the air density in units of g/cm^3. WC4 £ 0, default value of .001226 g/cm^3 is used (Dry air at 1 atm pressure and 15 deg C).</i>

Wind Table 2. Wind Formulas used in RMA2.

Option	Formula Name	Formula
1	RMA2 Original Formula	$T_S = (3.8E-6) \cdot W^2$ where T_S = wind stress (lb/(ft·sec ²)) W = wind speed (miles/hour) at 10 meters anemometer height *
2	Van Dorn Formula	$T_S = \rho_{ow} \cdot (A^2 \cdot W^2 + B^2 \cdot (W - W_{crit})^2)$ where T_S = wind stress (g/(cm·sec ²)) W = wind speed (cm/sec ²) at 10 meters anemometer height * ρ_{ow} = water density (g/cm ³) A = empirical Alpha coefficient (smooth water) B = empirical Beta coefficient (rough water) W_{crit} = critical wind speed for wave formulation (m/sec)
3	Wu Formula	$T_S = \rho_{oa} \cdot C \cdot W^2$ where T_S = wind stress (g/(cm·sec ²)) W = wind speed (cm/sec) at 10 meters anemometer height * ρ_{oa} = air density (g/cm ³) C = wind stress coefficient
4	Safaie Formula	$T_S = (\rho_{oa} \cdot \eta \cdot g) / A$ where T_S = wind stress (g/(cm·sec ²)) ρ_{oa} = air density (g/cm ³) η = dynamic roughness (cm) A = Charnock's constant (default is 0.0332) g = acceleration due to gravity (cm/sec ²)
5	Ekman Formula	$T_S = (3.2E-6) \cdot W^2$ where T_S = wind stress (g/(cm·sec ²)) W = wind speed (cm/sec) at 10 meters anemometer height *
6	Generic Formula	$T_S = \rho_{oa} \cdot C \cdot W^{exp}$ where T_S = wind stress (g/(cm·sec ²)) W = wind speed (cm/sec) at 10 meters anemometer height * ρ_{oa} = air density (g/cm ³) C = wind stress coefficient exp = empirically derived exponent
7	Van Dorn Formula with defaults	$T_S = \rho_{ow} \cdot (A^2 \cdot W^2 + B^2 \cdot (W - W_{crit})^2)$ where default values are W = 10 cm/sec wind speed (cm/sec ²) at 10 meters anemometer height * ρ_{ow} = 1.0 g/cm³ water density (g/cm ³) A = 1.1E-3 empirical Alpha coefficient (smooth water) B = 1.5E-3 empirical Beta coefficient (rough water) W_{crit} = 5.60 m/sec critical wind speed for wave formulation
8	Wu Formula with defaults	$T_S = \rho_{oa} \cdot C \cdot W^2$ where default values are W = 10 cm/sec wind speed (cm/sec) at 10 meters anemometer height * ρ_{oa} = 0.001226 g/cm³ air density (g/cm ³) C = wind stress coefficient (<i>see Wind Table 3</i>)

* Wind speed and direction are specified on the BW card.

Wind Table 3. Calculations for Default Wind Stress Coefficient.

Wind speed	Wind Stress Coefficient, C =
Wind Speed Less than 100	$1.25 / ((\text{Wind Speed}/100)^{0.2}) \cdot 0.001$
$100 < \text{Wind Speed} < 1500$	$((\text{Wind Speed}/100)^{0.5}) \cdot 0.001 / 2$
Wind Speed Greater than 1500	0.0026



See also: “Applying Wind Friction” on page 85, BW card.

BWS Card: Boundary, Wind Storm

Version 4.20 or higher.

Optional

EXPERIMENTAL

Card Description: Specifies information used to numerically describe storms or frontal passages occurring in the area to be simulated. Each BWS card describes an individual storm. You may simultaneously simulate multiple storm passages (presently dimensioned to 2) to describe complex wind patterns.

Field	Variable	Value	Description
0, C 1-2	IC1	BW	Card group identifier.
0, C 3	IC3	S	Card type identifier.
1	ISTYPE		Storm reference point (1-5)
		1	The location of maximum winds along the positive minor axis.
		2	The location of maximum winds along the negative minor axis.
		3	The center of the storm.
		4	The location of maximum winds along the positive major axis.
		5	The location of maximum winds along the negative major axis.
2	NREF	+	Reference node for storm track. (for ISTYPE)
3	TREF	+	Reference RMA2 simulation time for storm to arrive at node NREF.
4	SSPD	+	Storm tracking speed, mph.
5	WNDMAX	+	Maximum wind speed, mph (the speed at reference points 1, 2, 4, and 5).
6	WNDMIN	+	Minimum or base wind level, mph (the speed at reference point 3).
7	THETAS	+	Direction of storm track, degrees counterclockwise from positive x-axis toward which the storm is moving.
8	THETAK	+	Orientation of storm (long axis of ellipse) from (0, 180 deg) relative to x-axis in degrees counterclockwise.
9	BETAL	+	Standard deviation of spatial distribution function in the direction of the storm path.
10	BETAT	+	Standard deviation of spatial distribution function in the direction transverse to the storm path.
11	DECAY	+	Exponential temporal decay for wind speed (hr).

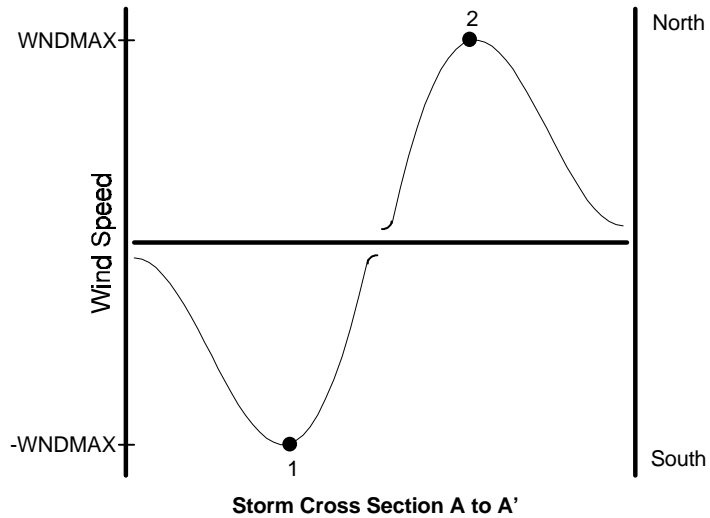
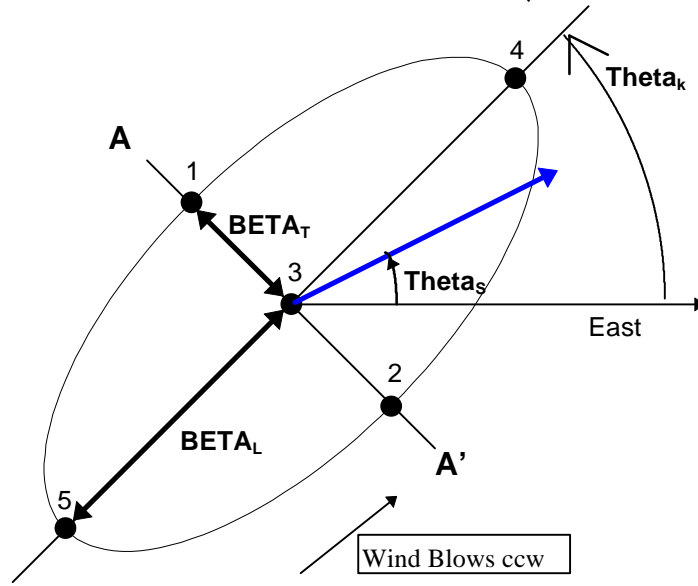


Tip: If BETAL is set much larger than the dimension of your mesh, and BETAT small, then a frontal passage can be simulated. Set $BETAL \gg BETAT$, and $BETAL \gg \text{mesh size}$.



See also: “Simulating With Storms” on page 86, “Applying Wind Friction” on page 85.



Storm variable explanation.



CA Card: Special Calculation Variables

Optional

Card Description: Used to specify modifications to default RMA2 calculation variables.

Field	Variable	Value	Description
0, C 1-2	IC1	CA	Card group identifier
1	USERCA1		Variable to calculate temporal derivative.
		-, 0	1.6 = default used if USERCA1 ≤ 0 or if no CA card is supplied.
		+	Code values between 1.0 and 1.6.
			 Note: A value of 1.5 for USERCA1 will specify a second order Taylor Series expansion.
			 Tip: Generally, values of USERCA1 which are closer to 1.0 will provide more model stability with less accuracy, while values closer to 1.6 provide more accuracy with less stability. With small time steps, the value of USERCA1 has little or no effect. With larger time steps, try values closer to 1.0 if model stability is a problem.



See also: “Adjusting RMA2 Calculations” on page 90.

CO Card: Comments

Optional

Card Description: Used to provide comments in the run control input file.

Field	Variable	Value	Description
0, C 1-2	IC1	CO	Card group identifier
1 - 10	FLD	Text	Any alpha-numeric data



Note: Comments may be supplied on this card anywhere within the run control file except as the first or last card types.







See also: “Any Comments?” on page 42.

DE Card: Wet/Dry by Elemental Elimination

Optional

Card Description: Used to specify information for the Elemental Elimination wetting and drying option. The original wetting and drying method for RMA2, this simplistic technique dries the *entire* element if any one node on that element has a computed depth less than the value specified with the DSET parameter, and rewets the element when all nodes have a computed depth greater than or equal to DSETD.

Field	Variable	Value	Description
0, C 1-2	IC1	DE	Card group identifier.
1	DSET		<p>The depth below which nodes are dry.</p> <p> Note: If the values specified on the TI card for the steady or unsteady state convergence criterion are less than zero, then DSET has a secondary purpose. It is involved in the criterion for sufficient depth convergence to allow the code to advance to the next dynamic time step <i>before</i> the iteration counter variable (NITN) on the TI card is satisfied. If the maximum depth change is less than ½ of DSET, the code advances to the next dynamic time step.</p> <p>0 Default value: 0.275 ft (or 0.084 m).</p> <p>+ User defined depth.</p>
2	DSETD		<p>The depth above which nodes become active when re-wetting.</p> <p>0 Default value: 0.60 ft (or 0.183 m).</p> <p>+ User defined depth.</p>
3	LI		<p>Iteration frequency of testing for wetting and drying.</p> <p>0 (Default) Prevents wet/dry testing.</p> <p>+ Wet/dry checking will be done at iterations on the specified interval.</p> <p>Typically, LI is a positive integer which is a multiple of the iteration counter (See NITI and NITN on TI card).</p> <p> Note: LI should never = 1.</p> <p> Tip: To permit a wet/dry check only at end of a time step, or revision of a time step (see REV card), set LI to a number larger than the iteration counter specified on the TI card.</p>

 **Note:** Default values are appropriately converted for SI units. If no DE Card is present, the default values are used.

**Example:**

DE .275 .6 4 *Wet/Dry checking will be done every 4 iterations.*



See also: “Elemental Elimination” on page 79, “Wetting And Drying” on page 78, DM card.

For details on the wetting and drying methods in RMA2, reference the following:


Roig, Lisa C., (1995) “Mathematical Theory and Numerical Methods for the Modeling of Wetland Hydraulics,” in Water Resources Engineering, Proceedings of the 1995 First International Conference, San Antonio, Texas, August 14 - 18, 1995, American Society of Civil Engineers, New York, pp. 249 - 253.

DM Card: Wet/Dry by Marsh Porosity

Version 4.25 or higher.

Optional

Card Description: Used to specify information for the Marsh Porosity wetting and drying option. The Marsh Porosity method makes a more realistic and *gradual* transition when wetting and drying than does the method of Elemental Elimination (DE card). All nodes on the element must be flagged dry before the element is considered dry.

Field	Variable	Value	Description
0, C 1-2	IC1	DM	Card group identifier.
0, C 3	IC3		Card type identifier.
		b (blank)	Specifies Option 1: Identified Node Option (IDNOPT).
		N	Specifies Option 2: Set wet/dry option for <i>node</i> J.
		E	Specifies Option 3: Set wet/dry option for <i>element</i> J.
		T	Specifies Option 4: Set wet/dry option for <i>material type (IMAT)</i> J.
1	J		Code J according to which option was chosen above with IC3 in data field 0, column 3...
			Option 1: Code J as one of the following (IDNOPT = J):
		0	Turns off the Marsh Porosity option. All DM cards are ignored.
		-1	RMA2 uses default values for data fields 2 through 5 for <i>all</i> nodes (user specified values are ignored).
		-2	User specifies the values in data fields 2 through 5 for <i>all</i> nodes.
		+	User specifies the values in data fields 2 through 5 for all nodes $\geq J$.
		+	Option 2: Code J as the <i>node</i> number. The values in data fields 2 through 5 will be used for this node.
		+	Option 3: Code J as the <i>element</i> number. The values in data fields 2 through 5 will be used for all nodes comprising this element.
		+	Option 4: Code J as the <i>material type (IMAT)</i> number. The values in data fields 2 through 5 will be used for all nodes comprising each element of this material type.
2	AC1 *	+	Distance from A0 to minimum regional bed elevation (Default = 3.0 ft or 0.91 m).
			 Note: A0 = average regional (nodal area) bed elevation (the 'z' value on GFGEN GNN card). If AC1 is ≤ 0 , see AC4 below.

The order in which DM cards should appear is: DMb (DM blank), DMT, DME, and finally, DMN. If a DMT card, for example, appears after a DME card, the DME card is potentially overruled by the DMT card..

3	AC2 *	+	Transition range of the distribution (Default = 2.0 ft or 0.61 m).
4	AC3 *	+	Minimum wetted surface area factor (Default = .02).
5	AC4 *	+	Minimum regional bed elevation. If $AC1 \leq 0$, $AC4 = \text{MIN}(AC4, A0 - \text{ABS}(AC1))$. If $AC4 = 0$, $AC1$ is used.

* See the figures below for definitions of parameters for fields 2 through 5.



Note: To include computations for the Marsh Porosity option, at least one DMb (DM blank) card is required, then optionally followed by DMT, DME, or DMN cards (in that order). If a node receives multiple assignments, the last assignment is processed.

The values for AC1 through AC4 are used as an “initial guess” for RMA2.

If an element becomes completely wet, DM card information for all nodes comprising that element is ignored until the water depth of the element falls within the transitional range and begins to dry.

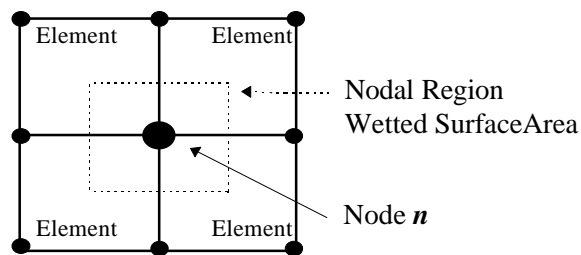
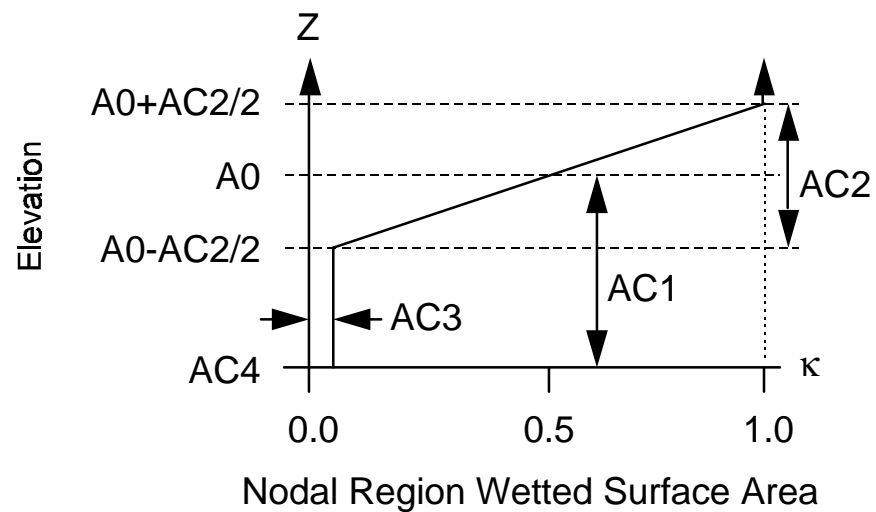
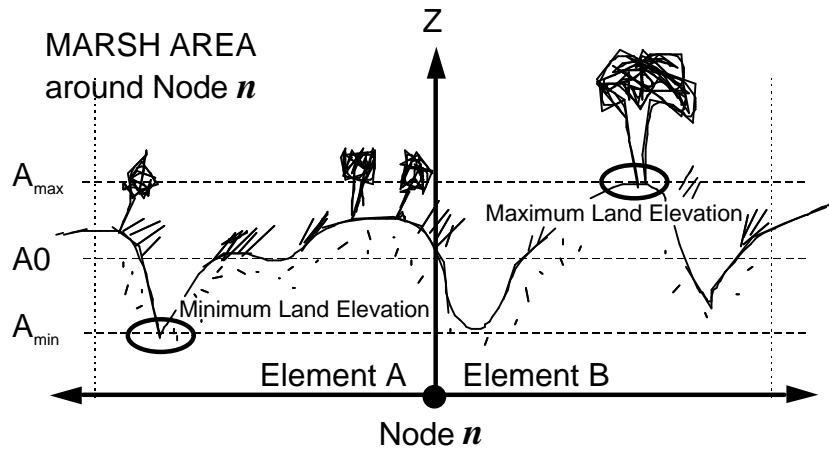
Default values are automatically converted to metric if the SI card indicates System International units.



See also: “Wetting And Drying” on page 78, DE card.

For details on the wetting and drying methods in RMA2, reference the following:

Roig, Lisa C., (1995) “Mathematical Theory and Numerical Methods for the Modeling of Wetland Hydraulics,” in *Water Resources Engineering, Proceedings of the 1995 First International Conference, San Antonio, Texas, August 14 - 18, 1995*, American Society of Civil Engineers, New York, pp. 249 - 253.



Where:

- A0** = average nodal area bed elevation ('z' value from GFGEN GNN Card). **A0** is the mean land elevation in the vicinity of node **n**.
- AC1** = distance from **A0** to minimum regional bed elevation.
- AC2** = transition range of the distribution.
- AC3** = minimum wetted area of the distribution.
- AC4** = minimum regional bed elevation.

*“Regional” refers to the “nodal area”; the area in the immediate vicinity of node **n**.*

END Card: End Card

Required

Card Description: Used to signal the end of all run control instructions for a given time step. For a steady state simulation, only one END card is required. For a dynamic simulation, there will be as many END cards as there are time steps.

Field	Variable	Value	Description
0, C 1-2	IC1	EN	Card group identifier.
0, C 3	IC3	D	Card type identifier.
1-10	FLD	Any	May be used for comments.



See also: “Specifying Boundary Conditions” on page 29.

EV Card: Turbulent Exchange Coefficients and Roughness

Required if a PE card, RD card or HN card, or, an EX card and EY card combination is not used

Card Description: Used to manually supply turbulence in the x and y directions, as well as a roughness value for the specified material type.

Field	Variable	Value	Description
0, C 1-2	IC1	EV	Card group identifier.
1	J	+	Element material type number (IMAT) for the set of turbulent exchange coefficients.
2	ORT(J,1)	+	E_{xx} = the x -momentum of turbulent exchange in the x -direction (lb-sec/ft ² or Pascal-sec for SI-units)*.
3	ORT(J,2)	+	E_{xy} = the x -momentum of turbulent exchange in the y -direction (lb-sec/ft ² or Pascal-sec for SI-units)*.
4	ORT(J,3)	+	E_{yx} = the y -momentum of turbulent exchange in the x -direction (lb-sec/ft ² or Pascal-sec for SI-units)*.
5	ORT(J,4)	+	E_{yy} = the y -momentum of turbulent exchange in the y -direction (lb-sec/ft ² or Pascal-sec for SI-units)*.
6	ORT(J,5)	+	Manning's n roughness coefficient (or Chezy if ≥ 3.0).

* 1 lb-sec/ft² = 47.879 Pascal-sec.



Note: Turbulent exchange coefficients (eddy viscosity) and roughness should be coded for every element material type represented in the mesh.

Recall the formula for Peclet numbers (P), where P is recommended to range between 15 and 40.

$$P = \frac{r u dx}{E}$$

where:

Coefficient	English Units	Metric Units
r = fluid density	1.94 slugs/ft ³	1 grams/cm ³
u = streamwise velocity	fps	mps
dx = length of element in streamwise direction	ft	m
E = eddy viscosity	lb-sec/ft ²	Pascal-sec



Tip: In many cases it is required to Coldstart the RMA2 model with “large” values of eddy viscosity (i.e., about 500) to achieve convergence. The eddy viscosity may be incrementally decreased by using REV cards or a series of Hotstart runs.



See also: “Modeling Turbulence” on page 37, EX card, EY card, HN card, PE card, RD card.

EX Card: Turbulent Exchange Coefficient, X-Velocity

Required if EV card or PE card is not used

Card Description: Used to manually supply the momentum of turbulence in the x direction. These values are specified according to material type.

Field	Variable	Value	Description
0, C 1-2	IC1	EX	Card group identifier.
1	J	+	Element material type number (IMAT) for the set of turbulent exchange coefficients.
2	ORT(J,1)	+	Exx = the x -momentum of turbulent exchange in the x -direction (lb-sec/ft ² or Pascal-sec for SI-units).
3	ORT(J,2)	+	Eyy = the y -momentum of turbulent exchange in the y -direction (lb-sec/ft ² or Pascal-sec for SI-units).



Note: Values for Eyx and Eyy are coded on the EY card.



See also: “Modeling Turbulence” on page 37.

EY Card: Turbulent Exchange Coefficient, Y-Velocity

Required if EV card or PE card not used

Card Description: Used to manually supply the momentum of turbulence in the y direction. These values are specified according to material type.

Field	Variable	Value	Description
0, C1-2	IC1	EY	Card group identifier.
1	J	+	Element material type number (IMAT) for the set of turbulent exchange coefficients.
2	ORT(J,3)	+	Eyx = the y-momentum of turbulent exchange in the x-direction (lb-sec/ft ² or Pascal-sec for SI-units).
3	ORT(J,4)	+	Eyy = the y-momentum of turbulent exchange in the y-direction (lb-sec/ft ² or Pascal-sec for SI-units).



Note: Values for Exx and Exy are coded on the EX card.





See also: “Modeling Turbulence” on page 37.

FC Card: Flow Control Structures for 1D

Version 4.20 or higher.

Optional

Card Description: Used to specify parameters to simulate flow control structures such as dams, weirs, culverts, etc.

Field	Variable	Value	Description
0, C 1-2	IC1	FC	Card group identifier.
1	NJN	+	Flow controller identifier (IMATs ≥ 904).
2	NJT		Flow controller type...
		1	Point source of flow, i.e., pump, storm drain , etc. Conditions are flow out equals flow in plus source (FLZ3), and equal total head at each node of the control element.
		2	Flow is a <i>reversible</i> function of head loss across the structure, i.e., reversible weir, open lock, open culvert , etc. Conditions are flow out equals flow in, and flow $Q = FLZ3 + FLZ4 * (HN1 - HN2 - FLZ5)**FLZ6$ in the direction FLZ7.  Note: If $HN1 - HN2$ is negative the sign of the flow direction is reversed.
		3	<i>An irreversible type 2 flow.</i> Flow is an <i>irreversible</i> function of head loss across the structure, i.e., a flapped culvert . Conditions are flow out equals flow in, and flow $Q = FLZ3 + FLZ4 * (HN1 - HN2 - FLZ5)**FLZ6$ in the direction FLZ7.  Note: If $HN1 - HN2 - FLZ5 < 0$ then $Q = 0$.
		4	Flow is a function of water surface elevation, i.e., a dam, weir , etc. Conditions are flow out equals flow in, and flow $Q = FLZ3 + FLZ4 * (HN1 - FLZ5)**FLZ6$ in the direction FLZ7.
		5	An optional way of specifying a type 2 flow controller above. <i>Head loss</i> is a function of flow Q . Conditions are flow out equals flow in, and head loss $HN1 - HN2 = FLZ3 + FLZ4 * (Q)**FLZ6$ in the direction FLZ7.
		6	<i>An irreversible type 5 flow.</i> Flow structure is irreversible. The head loss is a function of flow Q when the head loss is positive. If the head difference across the structure is negative, the flow is set to zero. $HN1 - HN2 = FLZ3 + FLZ4 * (Q)**FLZ6$ when $(HN1 - HN2) > 0$. $Q = 0$ when $(HN1 - HN2) \leq 0$.
3	FLZ3		Base flow Q in cubic feet per second (cfs or cms).
4	FLZ4		Relational coefficient.
5	FLZ5		Reference elevation or head difference (ft or m).
6	FLZ6		Exponent.

7	FLZ7	Direction of flow (radians, counter-clockwise from the positive x -axis).
---	------	---

HN1 and HN2 are the water surface elevations at the first and second nodes of the control structure element.



Tip: When multiple flow control structures are defined in the same mesh, it is generally good practice to have at least one of these structures defined such that the flow, Q , is a function of head loss, ΔH (type 2 or 3), and one such that ΔH is a function of Q (type 5 or 6).

For cases where extremely small head losses are encountered, the types 5 and 6 flow controllers may be numerically more stable.



See also: “Using One Dimensional Control Structures” on page 62.

FD Card: Fluid Density

Version 4.00 or higher.

Optional

Card Description: Used to supply a fluid density for all nodes or a specified node.

Field	Variable	Value	Description
0, C 1-2	IC1	FD	Card group identifier.
0, C 3	IC3		Card type identifier.
		b (blank)	Specifies Option 1: Universal assignment for all nodes $\geq J$.
		N	Specifies Option 2: Assignment by individual node.
1	J		The node or starting node number as specified by IC3 above.
		+	Option 1: The <i>starting</i> node number.
		+	Option 2: The node number.
2	RON(J)	+	Fluid density at node J (slugs/ft ³ or kg/m ³ for SI units).

Densities for Fresh Water and Sea Water.

Type of Water	English Units slugs/ft ³	Metric Units kg/m ³
Fresh Water	1.935	998.46
Sea Water	1.990	1026.84



Note: If no FD card is present, the density of fresh water is used.



Example:

FD 1.99 Use the density of seawater.



See also: “Fluid Density” on page 77.

FT Card: Water Temperature

Version 4.20 or higher.

Optional

Card Description: Used to supply the average initial water temperature for the entire mesh.

Field	Variable	Value	Description
0, C 1-2	IC1	FT	Card group identifier.
1	TEMP	+	Average initial water temperature (degrees Celsius).



Note: If no FT card is present, 15 degrees Celsius is used.



See also: “Fluid Temperature” on page 77.



G1 Card: Geometry, General Geometry Parameters

Optional (No longer supported)

Card Description: This card has been replaced by the GS card, LA card, and BCC card. It provided a means to:

- Supply an average Latitude used for Coriolis force calculations
- Specify an x and y scaling factor for the mesh
- Specify the number of boundary conditions per time step to be found in the input

which are now done on the above mentioned card types.

Field	Variable	Value	Description
0, C 1-2	IC1	G1	Card group identifier.
1	OMEGA	+	Average Latitude in degrees, local to the study area, used in calculating the Coriolis forces. Default is OMEGA = 0 to turn off Coriolis.  Note: LA cards with the T option provide for Coriolis by IMAT
2	XSCALE	+	Scale factor for X-coordinates. Default is 1.0.
3	YSCALE	+	Scale factor for Y-coordinates. Default is 1.0.
4	NBX	0, +	Specifies the number of nodes with boundary conditions specified with BCN cards. Default is NBX=0 and program will self-count the BCN cards.  Note: This data field is no longer necessary for normal operation. It is only necessary when some old style input format. It is recommended to use the default value.



Note: If no G1 card is present then:

- Coriolis is not applied (unless an LA card is used)
- Coordinate scale factors are 1.0 (unless a GS card is used)
- BCN cards are self counted




See also: BCC card, GS card, LA card.

GC Card: Geometry, Continuity Check Line

Optional

This card is being phased out and will be replaced by the GCL card.

Card Description: The GC card is used to specify a line within the grid where the flow rate is of interest. GC lines may be used to specify the location of boundary conditions.

Field	Variable	Value	Description
0, C 1-2	IC1	GC	Card group identifier.
1	LMT(K)	+	Total number of nodes to be listed for the continuity check line described by this GC card.
2	LINE(J,K)	+	List the node numbers 1, 2, ..., total number of nodes specifying the check line.  Note: If there are more node numbers specifying a flow continuity check line than will fit in the data fields remaining on the current GC card, continue coding the remaining node numbers starting in data field 1 of the next GC card.

In general, code the nodes making up the continuity check line from right to left when facing downstream. Code corner nodes only.



Note: Specify the dominant boundary line (inflow or outflow) first since that line is used in calculating the percent of the total flow at all subsequent lines. Code corner nodes only.

The maximum number of flow continuity check lines that can be calculated is determined by the value of MCC, a parameter variable in the source code (generally about 100). The maximum number of nodes allowed for one check line is determined by the value of MCCN.

Code all lines in the same direction (i.e., right to left across the flow), otherwise, flow rate results may be mis-interpreted since some flow rates may be reported in the printout as positive and some as negative.



Tip: When running a steady state problem, this feature can be used to help verify that the model is providing acceptable results by comparing the flow rate results at the points specified with the corresponding flow rates from field data. The flow rate across the line will be reported in the full results listing file.




See also: “Checking For Continuity” on page 44, GCL card.

GCL Card: Geometry, Continuity Check Line

Version 4.30 or higher.

Optional

Card Description: The GCL card is used to specify a line within the grid where the flow rate is of interest. GCL lines may be used to specify the location of boundary conditions.

Field	Variable	Value	Description
0, C 1-2	IC1	GC	Card group identifier.
0, C 3	IC3	L	Card type identifier.
1	J	+	Continuity check line number described by this GCL card.
2-n	LINE(J,K)	+	List the nodes that define this continuity check line.  Note: If there are more node numbers specifying a continuity check line than will fit in the data fields remaining on the current GCL card, continue coding the remaining node numbers starting in data field 1 of the next GCL card.
n	End of List	-1	A node number of -1 is required to mark the end of the list of nodes that specify this continuity check line.



Example:

```
CO Line#      Node numbers...
GCL   1         10   11   12   13   14   15
GCL         16   17   18   19   20   21
GCL         22   23  -1
CO   End of check line #1.
GCL   2        100  101  102  109  107  99  -1
CO   End of check line #2.
```

In general, code the corner nodes making up the continuity check line from right to left when facing downstream. Code only corner nodes to define the line.



Note: Specify the dominant boundary line (inflow or outflow) first since that line is used in calculating the percent of the total flow at all subsequent lines. Code corner nodes only.

The maximum number of flow continuity check lines that can be calculated is determined by the value of MCC, a parameter variable in the source code (generally about 100). The maximum number of nodes allowed for one check line is determined by the value of MCCN. Code all lines in the same direction (i.e., right to left across the flow), otherwise, flow rate results may be mis-interpreted since some flow rates may be reported in the printout as positive and some as negative.



Tip: When running a steady state problem, continuity check lines can be used to help verify that the model is providing acceptable results by comparing the flow rate results at the points specified with the corresponding flow rates from field data. The flow rate across the line will be reported in the full results listing file.





See also: “Checking For Continuity” on page 44.


GE Card: Geometry, Element Connection Table


Optional


Card Description: The GE card is used to create or modify elements in the mesh using nodes specified on GN cards with the N option, or nodes available in the mesh with which an element can be attached or constructed.

Field	Variable	Value	Description
0, C 1-2	IC1	GE	Card group identifier.
1	J	+	Element number.
2-9	NOP(J,I)	+	Code up to 8 node numbers for element J, listed counterclockwise around the element, starting from any <i>corner</i> node.
10	IMAT(J)	+	Material type number for element J (optional, may be specified on GT card).
11	TH(J)	-, 0, +	Direction of eddy viscosity tensor in radians , counterclockwise from the x-axis (optional, may also be specified on GV card).  Note: For 1-D elements, the direction is automatically aligned with the orientation of the 1-D element

 **Note:** The element connection table will usually be provided by the GFGEN pre-processor and will reside on the logical unit for GFGEN Geometric Data (\$L card). If so, the GE card should be omitted from the RMA2 run control file unless small mesh revisions are required. Otherwise, code the Nodal Point-Element Connection Table.

 **Tip:** To effectively remove an element from the computational mesh, set the material type (IMAT) for the element to a value of zero (the element will appear to be land). This technique may be useful when troubleshooting problems in your mesh.

 **Tip:** Another way the GE card can be useful is when you need to test changes to the mesh but you do not want to commit to modifying the geometry file until you know the results of the changes. You may use the GE card, along with other geometry cards, in the RMA2 run control file to make these types of changes.


 **See also:** “Modifying Elements” on page 70, GN card, GT card, GV card, and GZ card.

GN Card: Geometry, Nodal Point Coordinates

Optional

Card Description: Used to add a node to the mesh or to modify the (x, y, z) coordinates of an existing node.

Field	Variable	Value	Description
0, C 1-2	IC1	GN	Card group identifier.
0, C 3	ISI	N	Card type identifier.
1	J	+	Node number.
2	CORD(J,1)	-, 0, +	x-coordinate at node J.
3	CORD(J,2)	-, 0, +	y-coordinate at node J.
4	AO(J)	+	Bottom elevation.
5 (1D)*	WIDTH	+	Channel width at zero depth for node J.
6 (1D)*	SS1	-, +	Left side slope at node J.
7 (1D)*	SS2	-, +	Right side slope at node J.
8 (1D)*	WIDS	+	Storage width associated with node J at zero depth.

 **Note:** This feature is primarily used for model verification purposes.

** Used for 1D nodes only. If you use FastTABS version 3.02 or later, it is preferred that data for fields 5 through 8 be supplied on GW cards with the N option.*



Note: The coordinate values read (CORD(J,1), CORD(J,2), and AO(J)) are multiplied by the appropriate scale factors, XSCALE, YSCALE, and ZSCALE from the GS card. Be sure to specify the coordinates in the appropriate units (feet or meters) as specified on the SI card.

It is possible to use the GN card without specifying data field 4 through 8 if only nodal position is to be modified.



FastTABS Note: As of this writing, FastTABS does not support the one dimensional parameters on the GN card. When writing the RMA2 data file, FastTABS may omit your one dimensional data from the GN card.



See also: “Modifying Nodes” on page 70, GW card, GE card, GT card, GV card, and GZ card.

GS Card: Geometry, Scale Factors

Version 4.30 or higher.

Optional

Card Description: This card applies scale factors for the *x*, *y*, and/or *z* coordinates.

Field	Variable	Value	Description
0, C 1-2	IC1	GS	Card group identifier.
1	XSCALE	+	Scale factor for X-coordinates.
2	YSCALE	+	Scale factor for Y-coordinates.
3	ZSCALE	+	Scale factor for Z-coordinates (bottom elevation).



Note: If no GS card is present then all scale factors are 1.0.



See also: “Resizing The Mesh” on page 72

GT Card: Geometry, Element Material Types

Optional

Card Description: Used to specify or modify element material types.

Field	Variable	Value	Description
0, C 1-2	IC1	GT	Card group identifier.
1	J	+	Element number.
2	IMAT(J)	0, +	Element material type number.
3-10		+	You may provide (J, IMAT(J)) sets of values.



Tip: To effectively remove an element from the computational mesh, set the material type (IMAT) for the element to a value of zero (the element will appear to be land).



See also: “Changing An Element’s Material Type” on page 71, “Removing Elements” on page 71.

GV Card: Geometry, Eddy Viscosity Tensor

Optional

Card Description: Used to specify or modify an element's eddy viscosity tensor.

Field	Variable	Value	Description
0, C 1-2	IC1	GV	Card group identifier.
1	J	+	Element number.
2	TH(J)	-, 0, +	Direction of eddy viscosity tensor (Radians, counterclockwise from the x -axis)

If desired, you may fill the GV card in complete element/direction sets, or continue on another GV card.



Note: RMA2 requires an eddy viscosity tensor for every element in the mesh. Normally, this information is specified with GE cards in the GFGEN geometry file where the default value is zero degrees. The eddy viscosity tensor can also be specified in the RMA2 run control file on GE cards.





See also: “Modeling Turbulence” on page 37, GE card.

GW Card: Geometry, One Dimensional Channel Width Attributes

Version 4.20 or higher.

Required for one dimensional nodes if GN cards with the N option are not used

Card Description: Used to only specify or modify *one dimensional* trapezoidal channel attributes at the node specified.

Field	Variable	Value	Description
0, C 1-2	IC1	GW	Card group identifier.
0, C 3	IC3		Card type identifier.
		b (blank)	Specifies Option 1: Universal assignment for all nodes \geq NODE.
		N	Specifies Option 2: Individual node assignment.
1	NODE		The <i>one dimensional</i> starting node or node number as specified by IC3 above.  Note: Enter one dimensional corner nodes and transition nodes.
		+	Option 1: The <i>starting</i> node number.
		+	Option 2: The node number.
2	WIDTH	+	Channel surface width at zero depth for NODE.
3	SS1	-, +	Left side slope at NODE.
4	SS2	-, +	Right side slope at NODE.
5	WIDS	+	Storage bed width associated with NODE at zero depth.  Note: This feature is primarily used for model verification purposes.



Note: Code only *one* corner node per GW card.

If you are using FastTABS, the GW card is the preferred method for defining one dimensional channel width attributes as opposed to the GN card.



FastTABS Note: As of this writing, FastTABS will read and interpret, but will not update GW card data. Be aware that if the grid is renumbered by FastTABS, the node numbers on GW cards may no longer agree with the new element connection table. If this is the case, you will have to update the GW cards manually.




See also: “Using One Dimensional Elements” on page 59, GN card.


GZ Card: Geometry, Nodal Point Elevation


Version 4.30 or higher.


Optional

Card Description: Used to specify or modify the bottom elevation of the specified nodes.

Field	Variable	Value	Description
0, C 1-2	IC1	GZ	Card group identifier.
0, C 3	IC3		Card type identifier
		b (blank)	Specifies Option 1: A constant bottom elevation will be used for all nodes $\geq J$.
		T	Specifies Option 2: Bottom elevations by element material type (IMAT).
		N	Specifies Option 3: Bottom elevations by nodal point will be used.
1	J		The starting node or node number as specified by IC3 above.  Note: The nodes should be corner nodes.
		+	Option 1: The <i>starting</i> node number.
		+	Option 2: The element material type number.
		+	Option 3: The node number.
2	AO(J)	+	Bottom elevation.
3-10	etc.	+	Continue entering node number/elevation sets.

 **Note:** Code only corner node numbers. The bottom elevations are multiplied by the scale factor ZSCALE on the GS card. If desired, you may fill the GZ card in complete node/elevation sets, or continue on another GZ card.

 **Tip:** You can use GZ cards to create a dredged channel in your mesh. This technique is useful for making base/plan comparisons because you can leave the original geometry data intact. To remove the dredged channel, you only have to remove the GZ cards which define the channel bottom elevation from the RMA2 run control file.

 **Tip:** If you are experiencing problems diagnosing instabilities in an RMA2 simulation, you can use GZ cards to specify that all or part of the mesh has a flat bottom elevation. This technique may help pinpoint the trouble spot.

 **See also:** “Changing Bottom Elevations” on page 72.

HN Card: Roughness, Manning n-Value

Required if an EV card or RD card is not used

Card Description: Used to assign a Manning's n roughness value to the entire grid, or to an individual element material type.

Field	Variable	Value	Description
0, C 1-2	IC1	HN	Card group identifier.
0, C 3	IC3		Card type identifier.
		b (blank)	Specifies Option 1: Use a constant n-value for all <i>elements</i> $\geq J$.
		E	Specifies Option 2: Use a constant n-value for element J.
		T	Specifies Option 3: The n-value is specified by element <i>material type</i> (IMAT).
1	J		Code the element or material type as specified by IC3 above.
		+	Option 1 & 2: The <i>element number</i> .
		+	Option 3: The element <i>material type</i> .
2	ORT	+	Manning's n-value. If (ORT > 3) then the Chezy coefficient is used.



Note: It is advisable to always use an HN-*blank* card to set the n-value for all elements in the mesh. An EV card and/or RD card may then be used to override this n-value.



Tip: The table below suggests some Manning's n-values for various types of surfaces.

Manning's n-value Guidelines.

Manning's n-value	Condition
.015 - .020	Smooth earth with no weeds.
.020 - .025	Sand channel.
.034 (d_{50}) ^{1/6}	Rip-rap channels (d_{50} = the particle size of which 50% of the mixture is finer; i.e., 50% of the mixture is finer than this particle size).
.075 - .150	Very winding/overgrown.

Reference US Army Corps of Engineers publication EM 1110-2-1601 "Hydraulic Design of Flood Control Channels", Appendix J: "Methods for Predicting n-values for the Manning Equation"





See also: "Bed Friction And Resistance To Flow" on page 36, EV card, RD card.

IC Card: Initial Conditions

Required

Card Description: Used to provide initial conditions to start a cold run.

Field	Variable	Value	Description
0, C 1-2	IC1	IC	Card group identifier.
1	ELEV	+	Average initial water-surface elevation (feet or meters).  Tip: The most stable way to start a model run is to make sure that all nodes in the entire mesh are wet. In general, a <i>minimum</i> depth of about 2 feet on all nodes should be acceptable.
2*	HMIN	+	Minimum depth used for one dimensional nodes at startup.  Note: Eliminates possible negative depth on sloping river systems.
3*	UNOM	+	Nominal velocity for one dimensional nodes. Used as the initial guess if not restarting. Suggested value = .25 fps.

* Used for one dimensional elements only.



Note: If HMIN and UNOM are specified with the elemental wetting and drying option activated (DE card), then no wet/dry checks will occur in the first iteration.

HMIN and UNOM are unused for fully two dimensional networks.



See also: “Specifying Initial Conditions” on page 23.

LA Card: Local Latitude

Version 4.25 or higher.

Optional

Card Description: Used to specify the latitudinal position, north or south, of the area of interest, measured in degrees from the Earth's equator. This card provides information for the Coriolis Force correction calculation, which is used to account for angular velocity effects on bodies of water due to the Earth's rotation.

Field	Variable	Value	Description
0, C 1-2	IC1	LA	Card group identifier.
0, C 3	IC3	b (blank)	Specifies Option 1: Global assignment of average Latitude of the mesh by element.
		T	Specifies Option 2: Assigns specific Latitude by material type IMAT.
1	ISTART	+	Option 1: Code the first <i>element</i> number for which global assignment is to begin.
	IMAT	+	Option 2: Code the element <i>material type</i> .
2	OMEGA	+	Latitude in degrees. (Set OMEGA=0 to turn off Coriolis)

If desired you may fill the card with (IMAT, OMEGA) sets, or use more LAT cards. Latitude (Coriolis) may be *globally* assigned on the G1 card.



Note: Generally, the Coriolis Force correction should not be necessary unless you have a wide water surface (possibly > 100 miles) in the area of interest.



Warning: Instabilities on the southwest quadrant of the solution have been observed when using Coriolis Force correction. If this is a concern, a workaround is to apply graduated values of latitude (Coriolis). This is done by setting the latitude value for elements near the area of instability to zero, and increment the value of latitude in rows or layers of elements toward the point in the mesh where you want the value of latitude to be the true latitude. An increment of about 25% of the true latitude should be sufficient.





See also: "Compensating For The Rotation Of The Earth" on page 85, G1 card.

PE Card: Automatic Peclet Number Control of Eddy Viscosity

Version 4.28 or higher.

Optional

Card Description: Used to provide for *real time* adjustment of eddy viscosity based upon the computed velocity and individual size of each element. Larger elements and elements with higher velocities will have larger eddy viscosity values. Also, smaller Peclet numbers will result in larger values of eddy viscosity.

Field	Variable	Value	Description
0, C 1-2	IC1	PE	Card group identifier.
0, C 3	IC3	b (blank)	Specifies Option 1: Set Peclet parameters for the entire mesh.
		T	Specifies Option 2: Set Peclet parameters for the element <i>material type</i> (IMAT).
1	IPEC	0, 1	Option 1: IPEC is an On/Off switch for automatic eddy viscosity by Peclet number (Zero turns the option off).
	IMAT	+	Option 2: <i>Material type</i> for the Peclet parameter assignment.
2	GPEC	+	Peclet number.  Note: A smaller Peclet number will result in a larger value for eddy viscosity, and visa-versa.
3	VPEC	+	Coldstart: For every element of material type IMAT, VPEC is the <i>initial guess</i> used for the average elemental velocity. Hotstart: The <i>minimum</i> velocity (ft/sec, or meters/sec) to be used to compute the automatic eddy viscosity equation.  Note: Remember that if the SI card indicates Metric units, the velocity must be input in Metric units.
4	EPSXX	+	Scaling factor for computed eddy viscosity in the <i>xx</i> direction from the Peclet calculation (default = 1).
5	EPSXY	+	Scaling factor for computed eddy viscosity in the <i>xy</i> direction from the Peclet calculation (default = 1).
6	EPSYX	+	Scaling factor for computed eddy viscosity in the <i>yx</i> direction from the Peclet calculation (default = 1).
7	EPSYY	+	Scaling factor for computed eddy viscosity in the <i>yy</i> direction from the Peclet calculation (default = 1).



Tip: In many cases it is required to Coldstart the RMA2 model with “large” values of eddy viscosity (i.e., about 500) to achieve convergence. The eddy viscosity may be incrementally decreased by using REV cards or a series of Hotstart runs.

**Example:**

PE	1	20	1.0	1	1	1	1
PET	9	15	1.0	1	1	1	1

This example issues a global Peclet number of 20 as a global assignment everywhere, except a Peclet number of 15 will be used for material type 9 (IMAT=9).

Recall the formula for Peclet numbers (P), where P is recommended to be between 15 and 40.

$$P = \frac{r u dx}{E}$$

where:

Coefficient	English Units	Metric Units
r = fluid density	1.94 slugs/ft ³	1 grams/cm ³
u = average elemental velocity	fps	mps
dx = length of element in streamwise direction	ft	m
E = eddy viscosity	lb-sec/ft ²	Pascal-sec



See also: “Automatic Turbulence Assignment” on page 78.

RA Card: Rainfall And Evaporation

Version 4.297 or higher.

Optional

Card Description: Used to assign rainfall or evaporation at the specified elemental location or via material type association.

Field	Variable	Value	Description
0, C 1-2	IC1	RA	Card group identifier.
0, C 3	IC3		Card type identifier.
		b (blank)	Specifies Option 1: Use the unit discharge in data field 2 for all elements equal to or greater than J.
		E	Specifies Option 2: The <i>element</i> number for element inflow.
		T	Specifies Option 3: The element <i>material type</i> number for element inflow.
1	J		The element or material type number as specified by the value of IC3 above in data field 0, column 3.
		+	Option 1: The <i>starting element</i> number.
		+	Option 2: The <i>element</i> number.
		+	Option 3: The element <i>material type</i> number.
2	SIDF	-, 0, +	The elemental <i>inflow</i> in inches/hour (or cm/hr) Positive values represent rainfall (inflow), negative values represent evaporation (outflow).



Note : A positive value for SIDF represents rainfall, while a negative value represents evaporation.

Junction elements and other elements with material types greater than 900 cannot be assigned rainfall or evaporation.



See also: “Specifying Boundary Conditions” on page 29, “Adding Rainfall And Evaporation” on page 84.

RD Card: Automatic Roughness Coefficient Assignment by Depth

Version 4.28 or higher.

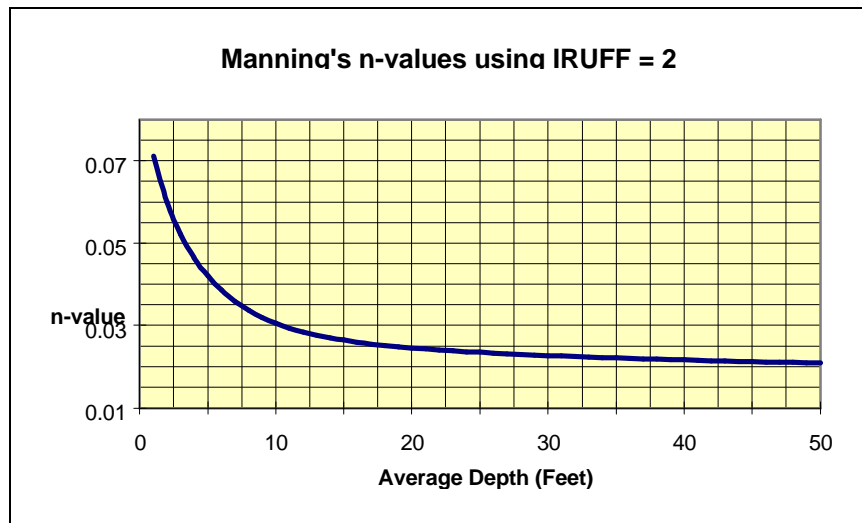
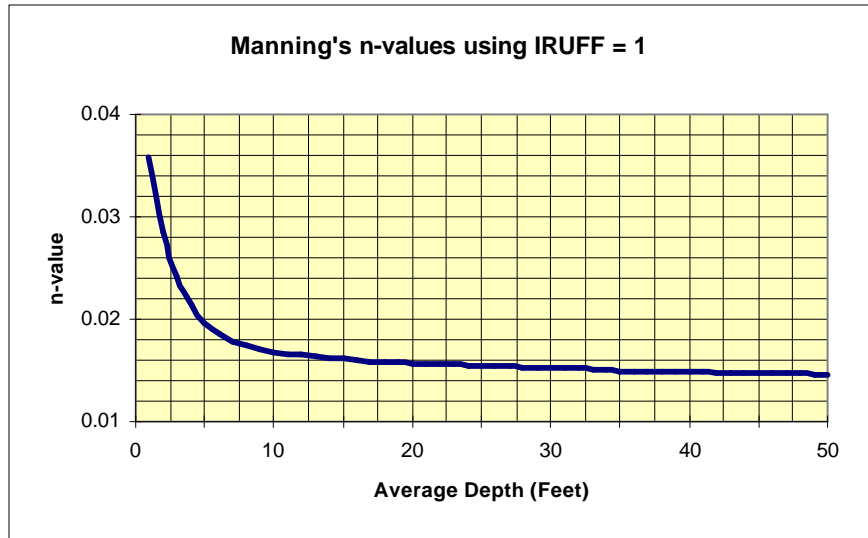
Optional

Card Description: Provides for real time adjustment of the bottom roughness coefficient. Automatically changes the Manning's n-value of an element depending upon the depth at each Gauss point. Generally, the deeper the water, the lower the roughness value.

Field	Variable	Value	Description
0, C 1-2	IC1	RD	Card group identifier.
0, C 3	IC3		Card type identifier.
		b (blank)	Specifies Option 1: Global assignment for element numbers \geq ISTART.
		T	Specifies Option 2: Set n-value by element <i>material type</i> (IMAT).
		E	Specifies Option 3: Set n-value by element <i>number</i> .
1	ISTART	+	If Option 1: Code the <i>first</i> element number for global assignment to begin. If Option 2: Code the element <i>material type</i> . If Option 3: Code the element <i>number</i> .
2	IRUFF	-, 0, +	On/Off switch for auto Manning's n-value calculations based upon an average depth (AVEDEP in the equation) of the element. A positive value turns the option on and sets defaults as in the table on the next page. A negative value or zero will turn the option off.
3	RDR0	0, +	Maximum Manning's n-value for non-vegetated water (a positive value will override the default).
4	RDD0	0, +	Depth at which vegetation effects roughness (a positive value will override the default).
5	RDRM	0, +	Manning's n-value for vegetated water (a positive value will override the default).
6	RDCOEF	0, +	Roughness by depth coefficient (a positive value will override the default).



Note: Default parameters are set as in the table on the next page depending upon the value of IRUFF.



$$NVALUE = RDR0 / (AVEDEP^{RDCOE}) + (RDRM * EXP(-AVEDEP/RDD0))$$

Default values for n-value parameters on RD card.

IRUFF	Project used for	RDR0	RDD0	RDRM	RDCOE
1	Miss River Delta project	.02	2.0	.026	.08
2	S-shaped River example test case	.04	4.0	.040	.166667
3	San Francisco Bay Estuary project	.04	2.0	.040	.166667



See also: “Varying Discharge During A Simulation (Rating Curve)” on page 74, EV card, HN card.

REV Card: Revise the Current Time Step

Version 4.25 or higher.

Optional

Card Description: Used to signal the RMA2 model to solve for the current time step. RMA2 will read data cards until it sees a REV card. At that point it will perform calculations and obtain an intermediate solution that will be used as *initial conditions* for the next series of data cards.

This card can be used as an alternative to Hotstarting. It can be used to gradually alter parameters to obtain the desired values.

Field	Variable	Value	Description
0, C 1-2	IC1	RE	Card group identifier.
1	IC3	V	Card type identifier.
2-10	FLD	Any	May be used for comments.



Example:

```
T3
$M
$L
.
.
.
EV  1  500  500  500  500  0.03
TI   4    0  .01
.
.
BQL  6 50000 -1.2  0.5
BHL  7   100
Solve for Hour 0.0 REV  Now lower coefficients and increase Q for 3 iterations
TI   3    0  .01
EV   1   300  300  300  300  0.027
BQL  6 75000 -1.2  0.6
Solve for Hour 0.0 given new data REV  Now lower coefficients more and increase Q
EV   1   150  150  150  150  0.025
BQL  6 100000 -1.2  0.7
Solves for Hour 0.0 and writes the END  Marks the end of the time step
solution using
EV = 150 and Q = 100000 and flow
distribution by depth = 0.7.
```



See also: “Revising Boundary Conditions During A Simulation” on page 73.

SI Card: System International Units

Optional

Card Description: Used to control the type of units RMA2 will use. Units are English or Metric.

Field	Variable	Value	Description
0, C 1-2	IC1	SI	Card group identifier.
1	METRIC	0	English units are applied (default).
		1	Metric units are expected as input and used for output.



Note: The SI card must be placed early in the card line-up in the run control file (immediately after the \$L card). If no SI card is present, English units are used.



See also: “Specifying Units” on page 67.

STO(P) Card: Stop the RMA2 Simulation

Required

Card Description: This card signals the end of *all* computation after the current time step has been completed.

Field	Variable	Value	Description
0, C 1-2	IC1	ST	Card group identifier.
1	IC3	O	Card type identifier.
2-10	FLD	Any	May be used for comments.



See also: “Stopping The Simulation” on page 40, END card.

T1-T2 Cards: Job Title

Optional

Card Description: Used to provide descriptive information about the data file.

Field	Variable	Value	Description
0, C 1	IC1	T	Card group identifier.
0, C 2	IC3	1, 2	Sequence.
2 - 10	TITLE	Any text	Any alpha-numeric data, up to 77 characters.



Note: A ‘T’ card must be the first user input card in the primary RMA2 run control file. Any number of T1 and T2 cards may be used and sequence is not significant. The title card section *must* be ended with a T3 card.



Example

```
T1      3700 Elements x 9115 Nodes x 20 Material types
T1
T2      Created by FastTABS - John Doe
T3      Mississippi River Gulf Outlet -- mean tide
.
.
```



See also: “Using Titles” on page 22, T3 card.

T3 Card: Job Title

Required

Card Description: Used to tell RMA2 that there are no more ‘T’ cards to be processed, and to provide descriptive information about the data file.

Field	Variable	Value	Description
0, C 1	IC1	T	Card group identifier.
0, C 2	IC3	3	Sequence.
2 - 10	TITLE	Any text	Any alpha-numeric data, up to 77 characters.



Note: A ‘T’ card must be the first user input card in the primary RMA2 run control file. Any number of T1 and T2 cards may be used and sequence is not significant. However, only one T3 card can be used and it *must* be the last title card in the set. RMA2 reads the ‘3’ to mean END of ‘T’ cards.



Tip: The alpha-numeric information on the T3 card is incorporated into the header of the RMA2 binary solution file. You should use this card to provide information that will later allow the solution file to be identified.



Example

```
T1      3700 Elements x 9115 Nodes x 20 Material types
T1
T2      Created by FastTABS - John Doe
T3      Mississippi River Gulf Outlet -- mean tide
.
```



See also: “Using Titles” on page 22, T1-T2 cards.

TI Card: Number of Iterations

Version 4.25 or higher.

Required

Card Description: Used to specify the iteration and convergence criterion for satisfactorily completing a time step.

Field	Variable	Value	Description
0, C 1-2	IC1	TI	Card group identifier.
1	NITI	0, +	The maximum number of iterations for the initial solution (steady state or Hotstart).
2	NITN	0, +	The maximum number of iterations for each dynamic time step(s).
3	SSDCRT	- 0, +	Uses DSET * .5 (from DE Card). User specified criterion for satisfactory depth convergence during steady state simulation (Default= 0).
4	USDCRT	- 0, +	Uses DSET * .5 (from DE card). User specified criterion for satisfactory depth convergence during an unsteady (dynamic) simulation (Default = 0).



Note: If you want the depth criterion to be based on 'DSET', the DE card must precede the TI card.



Tip: When Hotstarting, parameter NITI should be coded as zero only if no more iterations are desired for the initial conditions.



See also: "Iteration Control" on page 24, TZ card and DE card.

TO Card: Time for Saving the Binary Results Solution File

Optional

Card Description: Used to prevent RMA2 from writing to the solution file until the simulated hour matches a time specified.

Field	Variable	Value	Description
0, C 1-2	IC1	TO	Card group identifier.
1-10	TBINRY(J)	0, +	Time in decimal hours at which you want to save the final results to the binary file (Requires NOPT on the \$L card to be active). You may continue to list times until the card is full. If necessary, continue the list with another TO card.



Note: The decimal hour specified on this card must comply with the actual simulated hour (to five places past the decimal point).

If no TO card is specified, all time steps are written to the solution file.



Tip: Suppose you only need to save results at every simulated hour, but you want to make calculations at every simulated quarter-hour. You can use TO cards to specify the specific simulation hours you want to save in the solution file.



Example:

```
.  
. TO      25      26      27      28      29      30  
TO      31      32      33  
.  
.
```

In this example, results are written to the solution file starting at simulated hour 25, then continue to be written at every simulated hour until simulated hour 33.





See also: “When To Write To The Full Listing File” on page 68.

TR Card: Full Results Listing Control

Required

Card Description: Used to specify what information will be contained in the full results listing file.

Field	Variable	Value	Description
0, C 1-2	IC1	TR	Card group identifier.
1	IPRT		Control pertaining to startup conditions.
		0	Node and element input data is suppressed.
		1	Write all input data except initial conditions.  Note: Suggested when Coldstarting.
		2	Write all input data including initial conditions obtained from the results of the previous run (Hotstart).  Note: Suggested when Hotstarting.
2	ITSI		Control pertaining to nodal results. The number of iterations/time steps between successive writes.
		0	Do not print nodal results.
		1	All <i>iterations</i> are written.
		2	Every other <i>iteration</i> .
		3	Every third <i>iteration</i> .
		...	Etc...
		-1	Every <i>time step</i> (last iteration only).
		-2	Every other <i>time step</i> (last iteration).
		-3	Every third <i>time step</i> (last iteration).
		...	Etc...
3	IECHO		Switch to control HEC type card input data echo to terminal or log file.
		0	No input data echo (default).
		1	Run control input is echoed.
4 *	ITRACE		Trace subroutine calls and controllers.
		0	No trace.
		1-2	Degree of detail increases with the choice.

* Used for source code debugging purposes only.



See also: “Customizing The Full Results Listing File” on page 68.

TRN Card: Summary Results Listing Control

Optional

Card Description: Used to provide hydrographic information (velocities, depths, etc.) in a summary results listing file for the nodes specified.

Field	Variable	Value	Description
0, C 1-2	IC1	TR	Card group identifier.
C3	IC3	N	Card type identifier.
1-10	NSPLPT(J)	+	List of node numbers for the results listing summary. You may list nodes until the card is full. If necessary, continue the list with another TRN card.



Tip: Summary results listing data can be easily imported into a spreadsheet program to create charts and graphs, and to perform further analysis.



See also: “Requesting Only A Summary Of Nodal Results” on page 69.

TS Card: Timing For Binary Solution Output

Version 4.30 or higher.

Optional

Card Description: Used to control the time interval for writing time steps or iterations to the binary solution file.

Field	Variable	Value	Description
0, C 1-2	IC1	TD	Card group identifier.
1	IBHO		Scheme for saving RMA2 binary (u, v, h) solution (Default = -1).
		0	Never save the solution.
		+1	Save all <i>iterations</i> .
		+n	Save every n th <i>iteration</i> .
		...	etc.
		-1	Save results at the end of each <i>time step</i> .
		-n	Save results at the end of every n th <i>time step</i> .
		...	etc.
2	IBVO		Scheme for saving RMA2 binary vorticity solution (Default = 0).
		0	Never save the vorticity solution.
		+1	Save all <i>iterations</i> .
		+n	Save every n th <i>iteration</i> .
		...	etc.
		-1	Save results at the end of each <i>time step</i> .
		-n	Save results at the end of every n th <i>time step</i> .
		...	etc.



Note: It is advised to save only time steps in the solution file, not iterations. The ability to save iterations is intended for source code debugging and diagnostics.



See also: “Vorticity Solution File” on page 22, TO card.

TV Card: Iteration Control For Vorticity Calculations

Version 4.30 or higher.

Required if IVOR on the VO card is 1.

Card Description: Used to control the number of iterations for vorticity calculations.

Field	Variable	Value	Description
0, C 1-2	IC1	TV	Card group identifier.
1	NPASS1	+	The maximum number of passes between the steady state (u, v, depth) and vorticity iterations.
2	NVITI	+	The maximum number of steady state vorticity iterations.
3	NPASS2	+	The maximum number of passes between the (u, v, depth) and vorticity iterations per dynamic time step.
4	NVITN		The maximum number of dynamic vorticity iterations.
5	NGOODMAX	+	The number of good passes through a series of (u, v, depth) and vorticity calculations before the solution has converged for the time step. "Good passes" means the solution has converged before NVITI or NVITN is met.



Note: All RMA2 calculations will stop if NGOODMAX is achieved before NPASS1 or NPASS2 is reached.



See also: "Bendway Correction (Vorticity)" on page 74, BV card, VO card.

TZ Card: Computation Time

Required

Card Description: Used to control simulation time.

Field	Variable	Value	Description
0, C 1-2	IC1	TZ	Card group identifier.
1	DELT	+	Specifies the length of the computation time interval for a dynamic run, in decimal hours.
		0	Specifies that steady-state conditions will be used.
2	TMAX	0, +	The total run time in decimal hours.
3	NCYC	+	The total number of time steps (cycles).
4	NSTART		Specifies how many END cards to skip in the RMA2 run control input file before starting.
		0	Coldstart/Hotstart run. If REV cards are present in the <i>first time step</i> , then they will be processed.
		1	Coldstart/Hotstart run. Ignores any REV cards in the first time step.
		1+	Hotstart run. Skips through the RMA2 run control input file, keying on END cards, until it reaches the NSTART set. RMA2 then reads those boundary condition values to resume computations. For a Hotstart run, this should equal the last time-step of the prior run plus 1.
5	MBAND	0	Initial run or restarting on the first iteration in the convergence scheme.
		1	Restart at an intermediate iteration in the convergence scheme.



Note: RMA2 execution will stop whenever either TMAX *or* NCYC have been satisfied.



See also: “Total Simulation Time” on page 29, BCC card, TI card.

VO Card: Vorticity (Bendway Correction)

Version 4.30 or higher.

Optional

Card Description: Used to supply values associated with the calculation of vorticity.

Field	Variable	Value	Description
0, C 1-2	IC1	VO	Card group identifier.
1	IVOR	0, 1	On/Off switch for the vorticity (bendway correction) option. Vorticity is active if IVOR = 1. Default = 0.
2	SSVCRT	0, +	Steady state vorticity convergence criterion (refers to vorticity iterations only). Recommend: 0.00001. Default is .5 * DSET on the DE card.
3	USDVCC	0, +	Dynamic vorticity convergence criterion (refers to vorticity iterations only). Recommend: 0.00001. Default is .5 * DSET on the DE card.
4	ASEC	+	ASEC coefficient for the vorticity equation. Recommend: 5.0
5	DSEC	+	DSEC coefficient for the vorticity equation. Recommend: 0.5
6	RCMIN	+	Minimum radius of curvature that will be allowed. Recommend: 6 feet (or 2 meters).



See also: “Bendway Correction (Vorticity)” on page 74.

Performance Enhancements

If you have the RMA2 source code and a FORTRAN 77 compiler, you can customize RMA2 to better conform to your system. With a little experimentation and manipulation of array dimensions, you may be able to realize considerable performance improvements.

Why Is RMA2 So Slow?

RMA2 is a complex numerical model which performs many calculations per time step to obtain a solution. To run, it requires a system with ample resources (disk space, memory, CPU power). There are several factors which may contribute to a reduction in run performance.

Computer Processor Speed

Obviously, the faster your CPU, the faster RMA2 will be able to solve your problem. Unless you do not need results in a hurry, it is not recommended to run an RMA2 simulation of any significant size on a computer with less power than a 486 PC. In fact, a mainframe computer system or a workstation environment is the preferred setting.

Temporary Files

RMA2 writes temporary buffer files when the size of the solution matrix exceeds the dimensions of the matrix array. The temporary files are used as additional space to store the solution matrix. The number and size of these files depends upon the dimensions of the solution matrix array as defined in the source code, and upon the computer system on which RMA2 is running.

The process of writing to and reading from temporary files places a significant strain on the performance of RMA2, and is further exaggerated when the access time of the disk is slow. For information on reducing or eliminating the number of temporary files, see the section entitled "Reducing The Number Of Temporary Files" on page 170.

Disk Performance

The speed at which your disk can transfer data will have some influence on the time that RMA2 will take to complete your simulation. Especially if RMA2 is writing temporary files, a slow disk will kill performance.

Disks can become fragmented with use over time. Check your disk and de-fragment if necessary.

Fragmented disks can reduce performance. If your disk is fragmented, new files may be written in pieces and scattered all over the disk. This means the disk drive has to temporarily stop reading or writing the file, move to another location, and continue; thus, the loss of performance.

Reducing The Number Of Temporary Files

If your system has a large amount of RAM, say 32 Megabytes or more, it is likely you can improve performance by dimensioning RMA2 to decrease the number of temporary files that are written. You can reduce the number of temporary files by increasing the size of the buffer used to store the solution matrix. This is done by changing the variable NBS in the RMA2 include file to a value which is better suited for your system. For most problems, with the number of elements less than 10,000, a value between 100,000 and 200,000 should eliminate the temporary files.

Effects Of An Oversized Buffer

Some systems have the ability to use virtual memory, meaning the disk is used as memory when the available RAM is full. If you make the buffer size too large, RMA2 may run, but be aware that your system may run out of actual RAM and begin using the disk, in which case you are no better off than before, maybe worse.

The signs this is happening are that RMA2 is still running slow, the disk is constantly in use during the run, and there are no temporary files written. If this is the case, try reducing the buffer size until your system no longer needs to use virtual memory. The idea is to reach a compromise, having the least amount of temporary files while running entirely in RAM.

Redimensioning RMA2

RMA2 is written in standard FORTRAN 77 syntax.

There are several arrays whose size can be modified, and it should be determined by your problem and the system on which you are running RMA2. These array dimensions can be changed in the source code by editing the RMA2 include file and recompiling. The array dimension variables are as listed in the table below.

Variable	Description
MEL	Maximum number of elements in the mesh.
MND	Maximum number of nodes in the mesh.
MR1	Maximum number of equations. This is normally MND * 2.
MCC	Maximum number of continuity check lines.
MCCN	Maximum number of nodes per continuity check line.
MFW	Maximum front width. A function of the mesh domain length to width ratio, the number of nodes and elements, and the success of GFGEN's reordering (GO card).



FastTABS Note: The Mesh Information tool in FastTABS will provide a good estimate for the front width of your mesh.

MPB	Maximum size for the summary listing buffer. This should be large enough to accommodate the number of nodes specified on the TRN card, multiplied by the number of time steps. Example: To create a summary listing for 8 nodes, for a simulation with 40 time steps, MPB should be at least $8 * 40 = 320$.
MAXSTRM	Maximum number of storms simulated with BWS cards.
NBS	Maximum solution matrix buffer size (in RAM). This dimension influences the number of temporary files written.

Common Problems

Common Problems and Remedies

The Simulation Stops Prematurely

Cause 1

The Depth convergence parameter has exceeded a critical value of **25.0**.

Remedy

Examine the convergence parameters in the full results listing file to determine the first time step where divergence is noticed, then check for:

- negative depth at a node
- high velocities over shallow zones
- A Froude number beginning to approach one
 $\text{Froude \#} = \text{velocity} / \text{SQRT}(\text{gravity} * \text{depth})$
- large depth gradients around the zone
- performing a check for wetting and drying before the model has converged to a solution.
- irregular wet/dry patterns
- incompatible boundary conditions
- large boundary break angle
- abnormal Manning's n-value roughness, or eddy viscosity assignments
- non-smooth boundary condition signal
- delta time step assignment, etc.

If the problem isn't obvious, graphical tools are available.

Cause 2

The central processor unit has run out of allotted time. The simulation requires more CPU time than the computer allows. This problem can occur on mainframe type computer systems where a user's CPU use is limited.

Remedy

1. If possible, increase the maximum CPU time available to you.

2. Divide the simulation into two or more smaller simulations using the Hotstart capability of RMA2.
3. Use batch queues that offer larger CPU times.

Error Concerning the Temporary File (Logical Unit 9)

Cause 1

The \$M card has the wrong machine identifier.

Remedy

The machine ID = 1 for DOS PC, 3 for VAX, 4 for UNIX workstations, 5 for Cray Y-MP, and 8 for Macintosh.

Cause 2

All disk space has been exhausted.

Remedy

Delete any temporary files which may have been left from a previous RMA2 run. If there is adequate RAM available, recompile with a larger buffer size. See "Reducing The Number Of Temporary Files" on page 170.

An Element Has a Negative Area

Cause

The element connection was made clockwise rather than counter clock wise.

Remedy

Correct the corresponding GE card(s) in the GFGEN geometry file, or in the RMA2 run control file if the element is defined there.

An Element Has a Zero Area

Cause 1

An element with *zero* area may indicate that no eddy viscosity has been specified for the material type associated with the element.



Note: Junction and control structure elements have zero area by definition.

Remedy

Include an EV card for all material types which are used in the mesh.

Cause 2

Either the element does not exist in the network or the element material type has been set to zero (usually not fatal).



Note: A material type can be intentionally set to zero to remove the element or elements from the mesh. This technique is acceptable.

Remedy

Renumber the mesh.

Inconsistency In Units

Cause

The SI card metric flag does not match the GFGEN geometry.

Remedy

Verify which type of units you need to use and set the SI card appropriately in both the GFGEN geometry file and the RMA2 run control file.

Hotstart Difficulties

Cause 1

Improper settings for Hotstarting

Remedy

Proper setting are:

- NB > 0 on the \$L card
- IPRT = 2 on the TR card
- NITI = # on TI card (where # is zero if and only if no more iterations are desired for the initial condition)
- NSTART value set on the TZ card, where:
 NSTART = 1 if Hotstarting steady state, or
 NSTART = +1 more than the last time step count completed in the previous dynamic run.



Note: The program counts END cards, not REV cards, to determine how many boundary condition time steps to skip for a Hotstart run.

A Hotstart file is written only if the depth convergence maximum change is less than 2.0. A message is printed to the screen or logfile indicating when a Hotstart file is saved.

Hotstarting with an *old-style* alternate boundary condition file is not supported.

Cause 2

Geometry changes were made between Hotstart runs. The Hotstart immediately diverges.

Remedy

If any bottom elevation or other mesh editing has occurred between Hotstart runs, or some other abrupt change was added to the run control file, the old Hotstart file will no longer match the geometry and is useless. The simulation must be restarted from the beginning.

Cause 3

The initial conditions read from a Hotstart file generated from a previous RMA2 run do not match the first set of active boundary conditions. A diverged solution is the result.

Remedy

The NSTART value on the TZ card is wrong, or the last solution was not saved to a Hotstart file.

Exhausted All Disk Space

Cause

There is not enough space left on the disk for storing information.

Remedies

1. Purge unnecessary files or obtain more disk space.
2. Set variable ITSI to a large negative value to limit full print output (TR card).
3. Only save critical time steps (TO card).
4. Save the summary results listing rather than a full results listing (TRN card and \$L card).
5. Set the logical unit number to zero for all output files which are not critical (\$L card).
6. Check to see if all temporary files or core dumps were deleted.
7. If there is enough RAM available, set the buffer size dimension very large so the problem will run in memory rather than create temporary files.
8. Purchase more disk space.

Array Bounds Underflow

Cause

A node or element number is zero or negative.

Remedy

Examine the GFGEN geometry and/or RMA2 run control file.

Array Bounds Overflow

Cause

Too many nodes, elements, equations, continuity checks, etc. for the program dimensions.

Remedies

1. Decrease the problem size.
2. Increase the dimensions of the model.

Execution Terminated Because of 1D Nodes Lacking a Width Assignment.

Cause

No GWN card are present.

Remedy

Either re-run GFGEN with GN cards or GW cards with the N option to specify width, or add these cards to the RMA2 run control file.

Error While Reading Data

Cause 1

Unexpected end-of-file was hit before all data was processed.

Remedy

Check for:

- misspelled file names
- missing END cards and STO card
- wrong filenames assigned
- corrupted data file

Cause 2

An illegal card type.

Remedy

An illegal card type is typically a typographical or case error, or a blank line. Be sure the card names and parameters are typed correctly, and use upper case for card names.



Note: Never use the **TAB** key to separate data in RMA2 input files.



Note: If using an alternate boundary file, the first character of the file must be a 'B' for boundary or a 'C' for comment.

Error Reading An Input Binary File

Cause

The binary file must have been created by the same computer that is attempting to read the file. Transferring a binary file created from one computer to another computer which is not of the same family will not work! For example, from a Cray-YMP to a UNIX 32-bit workstation requires a binary conversion.

Remedy

Use a utility to convert the binary file to ASCII, move it to the alternate machine, then convert it back to binary on the new machine. See "Moving RMA2 Binary Solution Across Platforms" on page 195.

Execution Running Slower Than Expected

Cause

There can be many reasons for RMA2 to run slow. Below are a few suggestions for improving performance. See "Performance Enhancements" on page 169 for more information.

Remedy

1. If there is ample RAM available, try increasing the dimension of the parameter statement in the RMA2 include file which specifies the buffer size for temporary files. The idea is to have fewer buffer writes and more in memory.

2. Try to cut back on the number of files saved and the frequency of the write. Input/Output operations can severely decrease speed. Also, try to run the model at a location with high speed I/O.
3. Check to see if the disk space on the computer is fragmented.
4. When compiling the source code, use compiler optimizations.
5. Run the simulation when the computer is not sharing resources.



See Also: “Warning Messages” on page 179, “Error Messages” on page 182.

Warning And Error Messages

Warning Messages

AMW is Zero or Negative

Cause

This results from poor element shape or aspect ratio. Typically this is not fatal ... unless an element is flagged multiple times (16 maximum/element) and velocities become large.

Remedy

Edit the associated elements in the geometry to meet the suggested shapes. See the GFGEN Users Guide.

ARRAY Overrun In CRACK

Cause

Input parameters on a card go beyond column 80, or possibly a real or character value was read in an integer data field.

Remedy

Use a continuation card to code more data.

AutoPEC/N LU Turned Off Via \$L

Cause

These data fields are zero on the \$L card.

Remedy

Ignore, or stop and turn this feature on if desired.

BCN Card For Node= NODE' illegal

Cause

A new boundary condition assignment was attempted with a BCN card after a REV card or an END card, but this node number was not initially assigned with a BCN card for steady state.

Remedy

All boundary conditions must be initially defined for the steady state portion of the simulation. New boundary conditions cannot be added in mid simulation. Correct the run control file.

COEFS ASC (variable) near 0

Cause

Subroutine COEFS detects that the alignment of the boundary rating curve is near zero, and a division by this number could cause problems.

Remedy

Check the parameters on the BRC card.

Fear Of Dividing By Near Zero In COEFS

Cause

The water density value, ROAVG, is near or equal to zero.

Remedy

Be sure the water density value is specified properly on the FD card.

Final Binary Is Turned Off Via \$L

Cause

The \$L card NOPT variable is set to not save the final solution to a file.

Remedy

This is typically not done, since what you generally want from RMA2 is the solution. Ignore this warning, or if you need a solution, correct the NOPT variable on the \$L card.

Full Print Is Turned Off Via \$L

Cause

The \$L card IOUT variable is set to not save the full results listing file.

Remedy

Ignore this warning, or if you need a full results listing, correct the IOUT variable on the \$L card.

Geometry Was Not Defined Prior To xxx Card

Cause 1

GE cards were not read prior to the card in question in the RMA2 run control file.

Remedy

Change the card order RMA2 run control file by placing the \$L card and GE cards before the card in question.

Cause 2

The \$L card is set to not read the binary geometry.

Remedy

Change the IFILE variable on the \$L card so that the binary geometry is read.

Insufficient Information On xxx Card

Cause

All parameters were not completed on the card in question.

Remedy

Check the RMA2 run control file and correct the card.

IVRSID (Machine ID) = ?? Is Not Permitted

Cause

An invalid value for machine identifier was specified. The default value of 4 will be substituted

Remedy

Check the \$M card in the RMA2 run control file and provide a valid value which is correct for your machine type.

Node xxx Formed Dead End Without BC Specified

Cause

There is not a boundary condition assignment at the node specified.

Remedy

Specify a valid boundary condition for the specified node.

Reordering List Overruled If GFGEN Is Read

Cause

If a GO card is used in RMA2 (not recommended), it will overrule the reordering list that was read in by GFGEN.

Remedy

The use of a GO card is not recommended in RMA2. You are advised to reorder using GFGEN.

Storm Array Over-Run

Cause

You have requested or assigned more storms than the RMA2 is dimensioned to handle.

Remedy

Decrease the number of storms (BWS cards), or change the value of the MXSTRM variable to increase the number of storms RMA2 can simultaneously simulate.

Unused Elements

Cause

The element numbering is not sequential.

Remedy

Renumber the mesh, or proceed with caution.

Value of ISWTCH Invalid in CONVRT

Cause

Valid values are 1, 2, and 3.

Remedy

This is an internal program error. Please report via e-mail to **tabs@hl.wes.army.mil**.

You Just Turned Off xxxxxx

Cause 1

The \$L card is set to not read or write the file in question.

Remedy

Ignore the message, or change the \$L card.

Cause 2

The text “**null**” was entered at the interactive filename prompt.

Remedy

If you want to use this file, rerun RMA2 and enter the filename.



See Also: “Common Problems and Remedies” on page 173, “Error Messages” on page 182.

Error Messages

If the model is running interactively, an alarm beep or a screen flash will alert you that a problem has been detected. Most of these errors cause the model to stop execution.

Array Over-Run In Sub Check

Cause

Too many nodes were assigned on the stated continuity check line after the midside nodes were automatically inserted.

Remedy 1

Attempt to move the continuity check line to another location with less resolution.

Remedy 2

Increase the variable MCCN in the source code and recompile.

Cannot Modify Control Structure Not Defined At Beginning

Cause

An attempt was made to revise a control structure which has not been initially defined.

Remedy

Define the control structure at the beginning of the run.

Cannot Rain In Junction Or Control Structure

Cause

Rainfall or evaporation was specified via RA card for a junction element or control structure element. These element types have zero area.

Remedy

Remove all specifications of rainfall and evaporation from the RMA2 run control file for any elements with material types greater than 900.



See Also: “Adding Rainfall And Evaporation” on page 84.

Card Input Complete With xxx Errors

Cause

If any errors have occurred, the RMA2 simulation will stop after the initial run through and report the number of errors encountered.

Remedy

Tend to the errors and rerun.

Card xxx Affects A 1D Element That Lacks A Width

Cause

An RA card was applied to a one dimensional element which lacks a width assignment.

Remedy

Use a \$L card to read the geometry prior to the RA card, or use a GW card with the N option to assign a width to at the one dimensional node.

Card xxx Continuity Line Number xxx Has A 1st Node xxx

with an undefined width xxx or bogus bottom elevation

Cause

This error occurs when using Boundary Reflection and Absorption. The BRA type card works on one dimensional elements, and uses a continuity line assignment.

Remedy

You must define the one dimensional element prior to using the BRA card. Use a GC card, GN card with the N option, and/or GW card with the N option to define the element.

Card xxx Out Of Bounds Continuity Check Line

Cause

A continuity check line on the indicated card was determined to be negative, zero, or larger than the maximum number of continuity check lines RMA2 is dimensioned to deal with.

Remedy

Check for a typographic error, decrease the number of continuity lines, or increase the MCC parameter (maximum number of continuity check lines) in the source code and recompile.

Card xxx References An Out Of Bounds Element

Cause

An element on the indicated card was determined to be negative, zero, or larger than the maximum number of elements RMA2 is dimensioned to deal with.

Remedy

Check for a typographic error, decrease the mesh size, or increase the MEL parameter (maximum number of nodes) in the source code and recompile.

Card xxx References An Out Of Bounds Material Type

Cause

An element material type on the indicated card was determined to be negative or larger than the maximum number of elements RMA2 is dimensioned to deal with.

Remedy

Check for a typographic error, decrease the mesh size, or increase the MEL parameter (maximum number of nodes) in the source code and recompile.

Card xxx References An Out Of Bounds Node

Cause

A node on the indicated card was determined to be negative, zero, or larger than the maximum number of nodes RMA2 is dimensioned to deal with.

Remedy

Check for a typographic error, decrease the mesh size, or increase the MND parameter (maximum number of nodes) in the source code and recompile.

Depth Convergence Exceeds 25.0

Cause

The program stops and no solution is saved for the time step.

Remedy

See “The Simulation Stops Prematurely” on page 173.

If the problem isn’t obvious, graphical tools are available.

File Not Found

Cause

RMA2 cannot locate the specified file.

Remedy

Check to see that the filename is not misspelled, or that a wrong path was entered. Also, be sure the file is located where you believe it to be.

GC Card For Continuation Of Previous GC Was Expected

Cause

The continuity check line was not continued correctly. RMA2 expected a continuation GC card as a result of the corner node counter specified on the first GC card for the line.

Remedy

When using the GC(blank) Card:

For a given continuity line, the GC card contains a corner node counter as the first parameter, which tells RMA2 the number of corner nodes making up the continuity line. When the number of nodes is more than can fit on one GC card, a continuation card is required to continue the set. These continuation cards contain only the corner nodes and no counter value. There must be as many corner nodes coded as defined by the corner node counter on the first GC card for the line.

When using the GCL Card:

The GCL card contains only corner nodes (no counter parameter). The end of the node list is marked by a negative number. All continuation lines of for type of assignment must begin with GCL.

GFGEN Banner Shows Metric (or English) Units

(but SI card indicates English (*or metric*))

Cause

The SI card metric flag does not match the GFGEN geometry.

Remedy

Verify which type of units you need to use and set the SI card appropriately in both the GFGEN geometry file and the RMA2 run control file.

GFGEN Geometry Exceeds Program Dimension

Cause

The total number of node and/or elements in the geometry has exceeded RMA2 dimensions.

Remedy

Edit the source code and increase the variables MND for nodes and MEL for variables to appropriate sizes and recompile. Be aware that you must have enough RAM to load RMA2 after this change. See “Redimensioning RMA2” on page 170 for more information.

Illegal Card Type

Cause

The values contained in columns 1 through 3 in the RMA2 run control file did not reference a valid RMA2 data card.

Remedy

Check the typing and be sure you are trying to use a valid card type.

Length & Size Are Inconsistent With Dimension Capabilities

Cause

Subroutine CONVERT contains an error converting between integer and character variables.

Remedy

Internal program error. Please report via e-mail to tabs@hl.wes.army.mil.

Logical Unit # is Zero

Cause

Attempt to read non-defined alternate boundary condition input file. If the run control file is prematurely short, or contains no END card, RMA2 will switch to an alternate input file for more information.

Remedy

Specify that an alternate boundary condition input file is to be read with the IBUP variable on the \$L card, or put all boundary condition data in the initial run control file.

Next Input Record Is Too Long

Cause

The information on a data card exceeds 80 characters. The maximum amount of data allowed on a data card is 80 characters, including any spaces at the end of the line.

Remedy

Check the data card indicated and eliminate any unnecessary spaces.

NMAX is Not Sufficiently Large

(to permit the assembly of the next element ... Increase MFW parameter ...)

Cause

The mesh was not efficiently reordered in GFGEN

Remedies

1. Check the GFGEN GO card, and try other locations for ordering starting points.
2. Increase program dimensions of the front width.
3. Reduce the size of the mesh.

RMA2 Stops in Subroutine REVHYD ... Fatal Error

Cause

The continuity check line stated was not previously assigned by the stated card type at the beginning of the run.

Remedy

Correct the run control file by defining the continuity line.

RMA2 Has A Revise xxx Error

Cause

RMA2 cannot revise the card in question because it has not been initially defined.

Remedy

Correct the run control file by incorporating the card in question somewhere at the beginning of the run.

Side Slopes At Node xxx Non-zero

Cause

A one dimensional node was assigned a side slope, but RMA2 would not allow it here. For instance, a one dimensional transition node cannot have a left or right side slope.

Remedy

Remove any side slope specifications at the specified node.

Size Larger Than 1200

Cause

Array problem in subroutine CONVERT

Remedy

Internal program error. Please report via e-mail to tabs@hl.wes.army.mil.

“T” Card Expected, Run Terminated

Cause

The RMA2 run control file does not contain a valid Title card at the beginning of the file.

Remedy

Put a valid Title card at the beginning of the run control file.

The User Must Put The SI Card Before Specifying The Density

Cause

Misplaced SI card.

Remedy

The SI card should immediately follow the \$L card.

This File Is Mandatory

Cause

You cannot eliminate this file because it is required for the RMA2 simulation.

Remedy

Provide a valid file name when prompted, and do not attempt to “null”.

Too Many Continuity Check Lines

Cause

The number of continuity check lines specified via GC cards exceeds the RMA2 dimension for the maximum continuity check lines.

Remedy

Reduce the number of continuity check lines for your simulation, or increase the MCC parameter (maximum continuity check lines) in the source code and recompile.

Too Many Nodes In The Above Continuity Check Line

Cause

The number of nodes specified via GC cards per continuity check lines, plus the midside nodes which are automatically included, exceeds the RMA2 dimension for the maximum number of nodes per continuity check lines.

Remedy

If possible, break the indicated continuity check line into two individual lines using two GC cards, or increase the MCCN (maximum nodes per continuity check line) parameter in the source code and recompile.

Unsatisfied Elimination Error

Cause 1

The contents of LHED are ... (note the last equation number - EQN#)

NODE number ... NBC ARRAY IS

node # eqn#1 eqn#2 eqn#3 etc.

1. Element connection problem.

For example, two nodes having an identical (x, y) coordinate that are not one dimensional Junction or Control Structure elements.

2. Erroneous boundary conditions.

Example: specifying all water surface elevations with no inflows.

3. Fragmented mesh domain.

Part of the mesh has dried and caused ‘puddles’.

Remedy 1

Search the NBC ARRAY to match the EQN# echoed to determine which node number(s) are involved. Then closely examine the mesh in that area.

Cause 2

Subroutine FRONT cannot eliminate the element because its solution matrix pivot value is less than PIVOT_CRIT (default 1E-4). This may happen when all velocities are very near zero.

Remedy 2

Edit the RMA2 source code. Decrease the value of variable PIVOT_CRIT inside subroutine FRONT and recompile.



See Also: “Common Problems and Remedies” on page 173, “Warning Messages” on page 179.

RMA2 File Formats

RMA2 Binary Output File Format

All TABS-MD models operate on and generate binary files, each of which have several header or banner records that describe critical features about the contents of the file. The binary files are written by standard FORTRAN 77 unformatted write statements. The variable types follow standard FORTRAN assignments, where variables are REAL, except for those beginning with the characters 'I' through 'N' which are INTEGER. The contents of each record for the primary binary results file from RMA2 are described below.

Record 1

MFLG IREC NP NE

where MFLG - Model identifier flag
(120-129 for RMA2 results)
(135-139 for RMA2 Hotstart file)

IREC - Version number of the RMA2 program

NP - Number of nodes in the mesh

NE - Number of elements in the mesh

Record 2

IWRT1, (IBAN(i),i=1,IWRT1)

where IWRT1 - Number of items contained in the banner array

IBAN - Integer interpretation of the banner character strings

Record 3

IWRT2, IWRT3 , (IREC(i),i=1,IWRT2), (FREC(I),I=1,IWRT3)

where IWRT2 - Number of items contained in the IREC array

IWRT3 - Number of items contained in the FREC array

IREC - Integer flags which are set during execution

FREC - Floating point flags which are set during execution

Record 4

IWRT4, (ITIT(i),i=1,IWRT4)

where ITIT - Integer interpretation of the title character string
IWRT - Number of items contained in the ITIT array

Records 5 (Last)

TET, NP, ((VEL(j,k),j=1,3), k=1,NP), (NDRY(k), k=1,NP), NE,
(IMAT(k),k=1,NE), (WSEL(k), k=1,NP)

where TET - Simulation time, in decimal hours
NP - Number of nodes in the mesh
VEL - Array containing --> X-velocity, Y-velocity, and Depth
NDRY - Array containing wet/dry status for each node
(1 = wet, 2 = dry, -1 = About to become re-wet)
NE - Number of elements in the mesh
IMAT - Array containing each elements material type assignment
WSEL - Array containing water surface elevation for each node

Alternate Dynamic Boundary Condition Input File Format

An alternate file containing dynamic boundary information can be used in conjunction with the primary run control file. This file is useful when the dynamic boundary conditions do not change between several RMA2 runs. All steady state boundary data is supplied in the primary run control file.



Note: The use of an alternate dynamic boundary condition file is available but not supported.

To use an alternate file, be sure to

1. set IBUP = 61 on the \$L card
2. set ITSI = -# on the TR card
3. set DELT, TMAX, and NCYC on the TZ card
4. set NITN = 2 or more on the TI card
5. create an alternate dynamic boundary condition file, which may be in card style format or *old* format (described below)

The *old* style fixed format file is composed of line types A through I for each dynamic time step.

Line type A

1 line

Integer values end in columns 5, 10, 20, 25, 30, 35, 40, 45, 50.

NBX NSID DELT IQGEN IHGEN ISTGEN NCFLW IDEN IWIND

Line type B

IDEN lines, skip if IDEN = 0

Values end in columns 10 and 20.

NODE RON(NODE)

Line type C

NBX lines, skip if NBX = 0.

Values end in columns 10, 20, 30, 40, 50.

NODE NFIX(NODE) SPEC(NODE, 1) SPEC(NODE, 2) SPEC(NODE, 3)

Line type D

NSID lines, skip if NSID = 0.

Values end in columns 10 and 20.

IELEM SIDF(IELEM)

Line type E

IQGEN lines, skip if IQGEN = 0.

Values end in columns 10, 20, and 30.

ICONT QTOT THETA

Line type F

IHGEN lines, skip if IHGEN = 0.

Values end in columns 10 and 20.

ICONT HFX

Line type G

ISTGEN lines, skip if ISTGEN = 0.

Values end in columns 10, 20, 30, 40, 50, and 60.

ICONT AC1X AC2X AC3X AC4X THETA

Line type H

If IWIND = +, then provide NP lines, if IWIND = -, then provide 1 line, skip if IWIND = 0.

NODE TWX TAX

Line type I

NCFLW lines updating earlier controller data, skip if NCFLW = 0.

Values end in columns 10, 20, 30, ..., 70.

NJN NJTI FLZ3 FLZ4 FLZ5 FLZ6 FLZ7

Utilities

FastTABS

A graphical user interface for the TABS two dimensional numerical models. It provides a convenient means of building a mesh and viewing simulation results.



FastTABS is a proprietary product of Brigham Young University.

Moving RMA2 Binary Solution Across Platforms

Many TABS users work on multiple computer systems; PC's ,UNIX workstations, and mainframe computer systems. These different systems may manipulate binary files with a different word size. This makes it difficult to transfer a TABS binary solution file between these systems.

The solution to this dilemma is to convert the solution file to ASCII format, which can be transferred to any ASCII compliant computer. The new file is then copied to the second computer, and then converted again to binary on that system.

The utility programs **BIN2ASC** and **ASC2BIN** are available for this purpose. To be successful, you will need to compile both programs on both systems involved in the transfer.

Steps For Moving An RMA2 Binary Solution File

1. On the system where the solution file resides, run **BIN2ASC** to obtain an ASCII form of the solution.
2. Copy the ASCII file to the other computer system.
3. Run **ASC2BIN** on the new system to return to solution to a binary form compatible with the new system.

Subtracting Two RMA2 Solution Files

The utility program **R2DIFF** reads two RMA2 solutions created from the same geometry and subtracts the second from the first. The newly created RMA2 “look-alike” binary solution file can then be examined to view the differences between them. FastTABS provides a convenient way of viewing these differences. The areas with the most difference will be the most emphasized.

Merging RMA2 Solution Files

The utility program **MERGAVG** using the “merge” option, reads two or more RMA2 solutions created from the same geometry and merges the requested hours of simulation number 1 with simulation number 2, etc. The newly created RMA2 “look-alike” binary solution file can then be conveniently used for post-processing or as input to a TABS transport numerical model. This utility is useful when multiple dynamic Hotstarts were necessary.

This utility may also be used to exclude specified hours from a solution file by merging only one file and requesting only the hours you wish to save.

Averaging RMA2 Velocities

The utility program **MERGAVG** using the “average” option, reads one RMA2 solution and averages the velocities over the requested hours of the simulation. The newly created RMA2 “look-alike” binary solution file will be composed of only one time step with a designated hour of -99.00. It can then be conveniently used for post-processing or as input to a TABS transport numerical model.



Tip: In order to examine “net” or “residual” effects of velocity in an estuary, request the utility program to average over a tidal cycle from your simulation.

Obtaining Guideline Values Of Eddy Viscosity

The utility program **MAKE_EV** will read an ASCII geometry file and will, for each material type, determine the average size of the elements composed of that material type. The user is asked for a Peclet number he wishes to maintain, and, for each material type, a table is generated containing a range of velocities and associated eddy viscosity values which should maintain the given Peclet value (if the fluid is moving **S** fast, use the value for eddy viscosity associated with **S**).

Obtaining A Summary Listing After RMA2 Has Run

The utility program **R2_2_SUM** will create a summary results listing file from your solution after the RMA2 model simulation has run.

Technical Support

There are several technical support options available to TABS-MD users. We are continually striving to improve your support options to provide you with the answers you need as simply and as quickly as possible.

On-line Support

Technical support is available on the Internet 24 hours a day, 7 days a week.

Option 1: World Wide Web

Check out our list of Frequently Asked Questions on the Internet at URL <http://hlnet.wes.army.mil>.

Option 2: E-mail

If you cannot find a solution to your problem yourself, you can send us E-mail at tabs@hl.wes.army.mil. We will try to obtain a solution to your problem and reply with our findings.

TABS Hotline

Currently there is one person to handle Hotline support phone calls for TABS, so we request that you try all possible avenues to find a solution to your problem before calling for support. If you must call, please have the following pertinent data available:

- The GFGEN Geometry file
- The RMA2 run control file (.rc2 file)
- Computer information: type, memory capacity, etc.
- Any error messages
- Your E-mail and FTP addresses if applicable

The Hotline phone number is (601) 634-2730, FAX (601) 634-4208.

Glossary of Terms

Alternate boundary condition file

(Supplemental run control file)

A formatted input file which contains the boundary specifications for an RMA2 run.

Anemometer

A device used to measure wind speed and direction.



See Also:

BWC card.

ASCII

The **A**merican **S**tandard **C**ode for **I**nformation **I**nterchange 8-bit character set. ASCII values represent letters, digits, special symbols, and other characters.

An ASCII file, or text file, is a file which contains only ASCII characters in the range from 0 to 127.

Aspect ratio

An element's length to width ratio. Long, slender elements with an aspect ratio greater than 15 may cause stability problems.



See Also:

"Guidelines For " on page 16.

Banner

An alphanumeric set of information included in all TABS binary output files which describes the flow of data between GFGEN, RMA2, RMA4, and SED2D.

Base to Plan comparisons

The process of identifying differences in numerical model results between existing conditions and revised conditions, usually a change in geometry.



See Also:

“Verifying The Simulation” on page 43.

Batch mode

The opposite of Interactive mode. A job is typically submitted via proclv and is put in a job queue and will execute as CPU time and memory become available on the mainframe computer.



See Also:

“Running In Batch Mode” on page 16.

Bathymetry

The measurement of the depth of large bodies of water.

Binary

A numbering system consisting of only the numerals 0 and 1.

The TABS system uses some binary files. A binary file permits an efficient means to store numerical results.

Binary files are dependent on the word length of the computer from which they were created. They cannot be directly moved across computer platforms.

A TABS binary file is created by an unformatted FORTRAN WRITE statement.

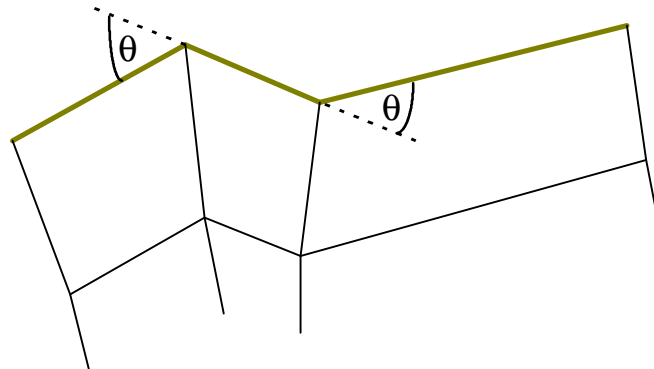


See Also:

“What Kind Of Computer Do You Have?” on page 23.

Boundary break angle

A boundary break angle is the angle at the node on the “land edge” boundary connecting two elements together. For RMA2 and RMA4, boundary break angles are recommended not to exceed approximately 10 degrees.



Boundary conditions

Water levels, flows, concentrations, stage/discharge relationships, etc., that are specified at the boundaries of the area being modeled. A specified tailwater elevation and incoming upstream discharge are typical boundary conditions.



See Also:

“Specifying Boundary Conditions” on page 29.

Boundary effect

A consequence of dissimilarities between the model boundary conditions and the conditions occurring in the prototype at the location of the model boundaries. This effect may be minimized if the model’s boundaries are far from the area of interest.



See Also:

“One Dimensional Elements” on page 5, and “Specifying Boundary Conditions” on page 29.

Boundary node

Any node which lies along an *exterior element* edge, or demarcates the wet/dry interface.

Buffer blocks

Temporary files created by RMA2, used as virtual memory to store the solution matrix when the BUFFER SIZE dimension value for the matrix arrays is not large enough to hold the matrix. If RMA2 runs to a normal completion, these files are automatically deleted.

Card

A term which comes from the 1960-1980’s when computers received data on punched cards. Each card supplied the computer with a line of data.

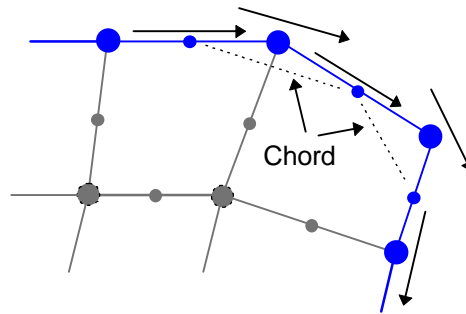
The TABS programs use cards in the same way. The difference is that the card data are stored in a disk file and not in a filing cabinet.

Card image

An ASCII line of data for the computer to read.

Chord

An imaginary line segment connecting two adjacent midside nodes. On a boundary, the flow direction for slip flow (parallel flow) is parallel to this chord.



Cold start

A model run using initial conditions that are not expected to be close to conditions as solved by the model, i.e., a level water surface elevation and velocity values of zero.

Collocated

Two or more objects placed together at the same location.

Compiler

A special computer program which converts a higher level language (such as FORTRAN) to a coded set of machine dependent instructions (fetch the contents of REGISTER 1). All TABS programs are written for the FORTRAN-77 compiler.

Continuation card

Used in a run control file. An additional card of the same type as the previous one, used to continue specifying input parameters.

Continuity

The term continuity refers to conserving mass within the model. The continuity check lines are typically used to estimate the flow rates and serve as an error indicator. The RMA2 model satisfied mass conservation in a weighted residual manner. The continuity lines can be used to check mass conservation a different way, by direct integration. Large discrepancies between the two methods indicate probable oscillations and a need to improve model resolution and/or to correct large boundary break angles.

Although continuity checks are optional, they are a valuable tool for diagnosing a converged steady state solution. For steady state, the continuity check lines should represent *total flow in equals total flow out*. However, if the continuity checks indicate a mass conservation discrepancy of $\pm 3\%$, you may want to address the resolution in the geometry. Large mass conservation discrepancies can lead to difficulty when the hydrodynamics are used for transport models, RMA4 and/or SED2D.

Continuity line

(or Continuity Check Line)

A string of corner nodes across which the total flow (or constituent if running RMA4) can be measured. The use of continuity lines also provides a convenient way to specify boundary conditions.

Control structure element

A special 2-node element with an IMAT ≥ 904 . The first node in the element connection table should be the side of the structure with the typical higher elevation.

See “Control Structure Elements” on page 8 for more information.

Convergence

The process of obtaining a solution by way of an iterative solution technique, such as the Newton-Raphson method.



See Also:

Diverge

Iteration

Convergence parameters

Related to the three RMA2 independent variables: x and y velocity components, and depth. The difference between previous and present values should approach zero with each successive iteration. If they do not, the solution is said to diverge.

For information on setting the convergence criterion, see “Providing A Convergence Criterion” on page 26.

Coriolis

Coriolis effect [for Gaspard Coriolis], tendency for any moving body on or above the earth's surface to drift sideways from its course because of the earth's rotational direction (west to east) and speed, which is greater for a surface point near the equator than toward the poles. In the Northern Hemisphere the drift is to the right of the motion; in the Southern Hemisphere, to the left. In most human-operated vehicles, continuous course adjustments mask the Coriolis effect. The Coriolis effect must be considered, however, when plotting ocean currents and wind patterns as well as trajectories of free-moving projectiles through air or water.



See Also:

“Compensating For The Rotation Of The Earth” on page 85

Corner node

Defines a vertex of an element. A point within the mesh that has an (x, y) coordinate and z depth.

If an element has three sides, then it has three corner nodes.

Curved boundary

An optional aesthetic means to outline key landmarks within the computational domain. A quadratic curved side is created by assigning (x, y) coordinates to the mid-side node of an element. Curving can help conserve mass in the transport models.



FastTABS Note: Curved boundaries may be created within the FastTABS program by unlocking the nodes and moving the mid-side node. Some automatic curving features are available as well.

Curved element side

An optional aesthetic means to outline key landmarks within the computational domain. A quadratic curved side is created by assigning (x, y) coordinates to the mid-side node of an element.

Data Field

A specific location on a record (card in TABS programs) in a data file where a data value occurs.

Datum

The horizontal plane to which soundings, ground elevations, or water surface elevations are referenced.

Some examples of a datum are:

- Mean high water
- Mean higher high water
- Mean low water
- Mean lower low water
- Mean sea level
- Mean tide level
- National Geodetic Vertical Datum

Delta time step

The increment of prototype time between two time steps.

Diurnal tide

A tide with one high water and one low water in a tidal day.

Diverge

The inability of the numerical model to achieve convergence by the iteration technique.



See Also:

“What Is An Iteration?” on page 24

Ebb tide

The period of tide between high water and the succeeding low water; a falling tide. Flow exits the estuary during ebb tide.

Eddy

A current of fluid moving contrary to the direction of the main current, especially in a circular motion. A whirlpool.

Eddy viscosity tensor

The eddy viscosity tensor defines the orientation (relative to the +x axis) of the primary longitudinal eddy viscosity, which is normally aligned with the primary flow direction. In other words, the E_{xx} (x momentum turbulent exchange in the x direction) is applied to the dominant or longitudinal direction of flow. For riverine cases, this angle may be set to the local river axis. This will then allow you to relax the transverse eddy viscosities. For estuarine conditions where it is difficult to define a primary flow direction, this variable is normally ignored and the transverse and longitudinal eddy viscosities are set the same.

Element

A segment, triangle, or quadrilateral shape composed of corner nodes and mid-side nodes. An element must be ‘connected’ to a neighboring element.

An element is composed of a list of nodes in a counterclockwise fashion and may define a 1, 2, or 3 dimensional problem. A line segment defines a one dimensional area, a triangle or quadrilateral defines a two dimensional area, while the three dimensional area is defined by adding layers to an element.

RMA2 can handle one dimensional and two dimensional elements.

Element connection table

The set of GE cards which define the nodes contained in each element

Exit boundary

A boundary condition location at which flow exits the mesh.

Far-field problems

Calculations at the boundaries are not considered important to the RMA2 simulation and are not to be used as accurate data for bankline information. Accuracy in the velocity field increases as you move away from the boundaries. For simulations where bankline conditions are of importance, a three dimensional model which addresses near-field problems should be employed.

FastTABS

A computer program that provides a graphical, point and click means for performing pre- and post-processing for surface water numerical models.

Developed at the Waterways Experiment Station and Brigham Young University. Corps of Engineers employees may contact the Waterways Experiment Station for more information via e-mail at tabs@hl.wes.army.mil, or call (601) 634-3339.

Fetch length

The distance across which the wind can blow without a land obstruction.

Field data

Data which has been collected at an existing, physical site, used when verifying the simulation.

Finite element

A method of solving the basic governing equations of a numerical model by dividing the spatial domain into elements in each of which the solution of the governing equations is approximated by some continuous function. This method lends itself well to the river/estuarine environments because of its diversity in computational mesh (element size, shape, orientation), flexibility of boundary conditions, and continuity of the solution over the area.

Flood Tide

The period of tide between low water and the succeeding high water; a rising tide. Flow enters the estuary during a flood tide.

Flow fields

The domain in which the water flows.

Free field

A data format where spaces or commas separate the data items on the input line. There is no fixed position in which the data is required to be located. Only the order of the items is important.

Free-surface flow

A fluctuating water surface elevation. A numerical model which can calculate a changing water surface elevation is a free-surface model. Models designated as “rigid lid” do not permit free-surface calculations.

Front width

The number of equations in the numerical model’s solution matrix that are assembled simultaneously.

Frontal passages

As in weather; refers to a meteorological storm front.

Froude number

A unitless mathematical expression used to describe a flow field. Froude numbers greater than or equal to 1 are supercritical, less than 1 are subcritical. RMA2 may become numerically unstable with Froude numbers greater than about .6.

The equation for Froude number is

$$F = \frac{V}{\sqrt{gh}}$$

where

F = the Froude number

V = the fluid velocity

g = gravity

h = fluid depth

Galerkin method

of weighted residuals

The Galerkin method of weighted residuals is a finite element method which requires that the integral of the residual times the weighting functions should equal zero.

Gate

A movable barrier, such as a tide gate, in a river or stream.

Gauss point

Sampling location within the element used for numerical integration. There are 9 Gauss points for a triangle and 16 Gauss points for a quadrilateral.

GFGEN

Geometry File GEneration program used to create the computational mesh for all TABS applications.

Gradient

The difference between the bottom elevation of any two corner nodes of an element. Large streamwise depth gradients along short distances are unstable.

Harmonic constituents

The component tides are usually referred to as harmonic constituents. The principal harmonic constituents of the tide are

Name of the component	Symbol	Period (hrs)
Principal lunar semidiurnal	M ₂	12.42
Principal solar semidiurnal	S ₂	12.00
Larger lunar elliptic semidiurnal	N ₂	12.66
Luni-solar diurnal	K ₁	23.93
Lunar diurnal	O ₁	25.82

The routine prediction of tides is based upon a simple principle which asserts that for any linear system whose forcing can be decomposed into a sum of harmonic terms of known frequency (or period), the response can also be represented by a sum of harmonics having the same frequencies (or periods) but with different amplitudes and phases from the forcing. The tides are basically such a system.

For the open coastal regions, the tidal prediction capability requires only that you have prior observations of the tides at the station over a suitable period of time from which the amplitudes and phases of the major harmonic constituents can be ascertained. There are typically less than 40 amplitudes and phases for important periods required to reconstitute a tidal signal. Fortunately there are computer programs available which can provide a predicted tidal signal at most every published USGS station location.

Head of tide

The location up river where the tidal signal has been damped so that it is insignificant.

HEC format

A naming convention for the style of run control input derived by Hydrologic Engineering Center (HEC) in which each line of input is defined by a 'Card Type in data field 0' and the data follows in fields 1 through n.

Example, GN card with the N option:

Field 0	Field 1	Field 2	Field 3	Field 4
GNN	node	x coordinate	y coordinate	bottom elevation

High tide

The maximum elevation reached by each rising tide.

Homogeneous fluid

A fluid which has uniform properties.

Hotstart

The process of supplying the numerical model a set of initial conditions which were obtained from the results of a previous simulation.



See Also:

“Hotstart File” on page 20 and “Resuming A Stopped Simulation” on page 64.

Hydraulics Laboratory

The US Army Corps of Engineers, Waterways Experiment Station, Hydraulics Laboratory, Vicksburg, Mississippi, is the principal Corps agency for engineering research and experimentation in hydraulics and hydrodynamics and is one of the largest hydraulics laboratories in the world.

The Hydraulics Laboratory provides TABS Numerical Model maintenance and support for Army Corps installations. To find out about availability and support for the TABS modeling system, contact the Hydraulics Laboratory via e-mail at **tabs@hl.wes.army.mil**, or call (601) 634-3339.

Hydrodynamic

Relates to the specific scientific principles that deal with the motion of fluids and the forces acting on solid bodies immersed in fluids, and in motion relative to them.

Hydrograph

A time series recording of the measurement of flow across a river or stream.

Hydrostatic assumption

Vertical pressure is balanced by gravitational forces.

$$\frac{\partial P}{\partial z} + \rho g = 0$$

where

P = pressure

z = water depth

ρ = density

g = gravity

ill-formed elements

Elements which do not conform to the rules for constructing a finite element mesh. See the GFGEN Users Guide for descriptions of ill-formed elements.

IMAT

Material Type. A variable name used in TABS programs to specify a number representing the type of material within an element. The IMAT is located on the GE card and is used to aid in assigning modeling coefficients.



Note: IMAT=0 is equivalent to assigning a land boundary around the element.

Inflow boundary

A boundary condition location at which flow enters the mesh.

Interactive mode

Opposite of Batch mode. The program requires the user to respond to questions.

If the program is running on a mainframe computer, the program is time sharing the CPU with other jobs, which can cause delays in some cases.

Iteration

Repeating a sequence of instructions a specific number of times, changing parameters and obtaining a new solution each time, until a predetermined condition is met.



See also:

“What Is An Iteration?” on page 24

“How RMA2 Finds A Solution” on page 24

Junction element

A special element, consisting of 3 to 8 nodes, which defines the intersection of 3 to 8 one dimensional elements.

Leaking

A description of the inability of a mesh to properly hold water. Some modelers refer to a ‘leak test’ as a means to check out a mesh.

Leaks, or “oozes”, are a result of poor element shapes, large boundary break angles, and/or erroneous boundary condition specifications.

Logical unit

Computer lingo used to associate a device number with a data file. In this FORTRAN statement, **10** is the logical unit number:

READ(10,*) DATA

Low tide

The minimum elevation reached by each falling tide.

Magnitude of velocity

A scalar value; the magnitude, M , of the resultant velocity vector. It is the square root of the velocity x component squared plus the y component squared.

$$M = \sqrt{x^2 + y^2}$$

Material type

A number representing the type of material within an element. The associated variable is IMAT.

Mean high water

(MHW) The average height of the high waters over a 19 year period.

Mean higher high water

(MHHW) The average height of the higher high waters over a 19 year period.

Mean low water

(MLW) The average height of the low waters over a 19 year period.

Mean lower low water

(MLLW) The average height of the lower low waters over a 19 year period.

Mean sea level

(MSL) The average height of the sea surface for all stages of the tide over a 19 year period, usually determined from hourly height readings.

Mean tide level

(MTL) A plane midway between mean high water and mean low water. Not necessarily equal to mean sea level.

Mesh

A collection of nodes and elements which defines the domain of the study area.

Mid-side node

A node between two corner nodes in an element. TABS models require mid-side nodes.

Mild Slope Assumption

In the derivation of the governing equations, the mild slope assumption is applied, which assumes that the normal force at the bottom of the bed is essentially vertical. This also results in the assumption that there is no horizontal component of that normal force to be accounted for in the horizontal momentum equations.

Mixed tide

A type of tide intermediate to those predominantly a semidiurnal tide and those predominantly a diurnal tide.

National Geodetic Vertical Datum

(NGVD) A fixed reference adopted as a standard geodetic datum for elevations determined by leveling. Established in 1929. Also referred to as National Geodetic Vertical Datum of 1929 and Sea Level Datum of 1929. The NGVD is usually preferred as the primary datum for engineering design.

Neap tide

A tide occurring approximately midway between the time of new and full moon. The neap tidal range is usually 10 to 30 percent less than the mean tidal range.

Node

A point containing an x , y , and z coordinate which defines a location in space. Mid-side nodes (x , y , z) are linearly interpolated from adjacent corner nodes, unless the element side is curved.

Null point

A location in a network where there is no net fluid transport (no flow).

Off-channel storage

A one dimensional element feature. The storage width associated with the node at zero depth, as specified on GN or GW cards using the N option.

One dimensional element

A line segment composed of two corner nodes and one mid-side node. The geometry is defined by cross section (a straight bottom line between corner nodes) and reach length. The calculated velocity is averaged over the cross section.

Parameter revision

The process of modifying a run control input parameter during a simulation in the middle of a time step. The REV card is used for this purpose.

Peclet number

Defines the relationship between element properties, velocity, and eddy viscosity.

$$P = \frac{\rho u dx}{E}$$



See Also:

“Modeling Turbulence” on page 37, PE card.

PREHYD

The subroutine within RMA2 that reads the TABS formatted run control file..

Proclv

The user friendly interactive procedure on mainframe WES Cray computers which permits users to access the library versions of the TABS system.



See Also:

Batch mode

Prototype

Field data or physical model data.

The original, or basis for the new study.

Record length

In this context, a FORTRAN specific term dealing with the size of a data type in a binary file.

Resource Management Associates

The TABS numerical models were initially developed by Dr. Ian King at Resource Management Associates, (RMA), in Lafayette, California. An RMA representative can be reached at (707) 864-2950.

Rip tides

A current of water disturbed by an opposing current. A strong, narrow current that flows rapidly away from the shore, returning water carried landward by waves.

Rip-rap

Stones, chunks of concrete, or other debris on an embankment slope, or stream side slope, generally used to prevent erosion.

RMA10

RMA10 is a multi-dimensional (combining 1-D, 2-D either depth or laterally averaged, and 3-D elements) finite element numerical model written in FORTRAN-77. It is capable of steady or dynamic simulation of 3-dimensional hydrodynamics, salinity, and sediment transport. It utilizes an unstructured grid and uses a Galerkin-based finite element numerical scheme. The WES-HL version is based upon the work of Dr. Ian King of Resource Management Associates.

RMA2

The one dimensional/two dimensional depth averaged hydrodynamic Finite Element numerical model within TABS.

RMA4

The one dimensional/two dimensional depth averaged water constituent transport Finite Element numerical model within TABS.

Roughness

In a river or stream bed, the material on the side slopes or the bottom, such as stones, etc., which inhibit the flow.



See Also:

“Bed Friction And Resistance To Flow” on page 36.

Run control file

An input data file which provides parameters that control the RMA2 simulation run.

SED2D

Originally known as STUDH, SED2D is a two dimensional depth averaged sediment transport Finite Element numerical model within TABS.

Semidiurnal tide

A tide with two high and two low waters in a tidal day with comparatively little diurnal tide inequality.

Ship simulator (WES)

The Waterways Experiment Station Navigation Simulator started out in 1984 as a single ship simulator with one view screen, a radar screen, a ship console, and a tow console running on a PDP-11 computer. The visual scene at that time was comparable to early computer games in color and detail. In the twelve years since then, the view screen has changed to three large screen monitors for a total viewing angle of 140 degrees. A second simulator has been added along with the capability of two-way traffic.

In the two-way traffic mode of operation, the two pilots interact vocally through an intercom, visually through the view screens, and physically through the hydrodynamic model controlling the reaction of the ship due to currents, wind, channel geometry, and traffic interaction.

In 1995 a new facility was built within the Hydraulics Laboratory at WES to house the two simulators and the new computer system that will run both simultaneously. The most recent addition to the Navigation Simulator was visual scene software incorporating hardware texturing and many other features to give the visual scene a more “realistic” look.

Source code

The US Army Corps of Engineers, Waterways Experiment Station, Hydraulics Laboratory, Vicksburg, Mississippi, provides TABS Numerical Model maintenance and support for Army Corps installations. To find out about availability and support for the TABS modeling system, contact the Hydraulics Laboratory via e-mail at tabs@hl.wes.army.mil, or call (601) 634-3339.

Special elements

Junction element, transition element, or control structure element.

Spin-up

The process by which a model moves from an unrealistic set of initial conditions to more realistic results that represent a solution that is not strongly influenced by the initial conditions.

To estimate spin-up, calculate the time for a wave to travel the length of the mesh and return.

The speed of the wave is calculated as

$$S = \sqrt{gh}$$

where

S = wave speed

g = gravity

h = depth

Spring tide

A tide that occurs at or near the time of new or full moon and which rises highest and falls lowest from the mean sea level.

Steady state

A simulation in which the boundary conditions are static. The variables being investigated do not change with time. RMA2 considers the steady state simulation time as hour zero.

Storm passages

A significant meteorological event that passes over the domain of the model.

STREMR

A depth averaged finite difference hydrodynamic numerical model developed by Bernard and Schneider of WES.

Subcritical

Froude number < 1

Supercritical

Froude number ≥ 1

System International

(SI) Formally named in 1960 by an international general conference on weights and measures. This system provides exact definitions of the metric system units for the fields of science and industry.

TABS

The TABS-MD Modeling System is comprised of four main programs: GFGEN, RMA2, RMA4, and SED2D.

Tailwater

The water surface elevation at the exit boundary.

Thalweg

The line representing the deepest part of a river or channel.

Tidal day

The time of the rotation of the earth with respect to the moon, approximately 24 hours and 50 minutes.

Tidal range

The difference in height between consecutive high and low (or higher high and lower low) waters.

Tide

The periodic variation in the surface level of the oceans and of bays, gulfs, inlets, and estuaries, caused by gravitational attraction and relative motions of the moon and sun.

The types of tides are:

- Diurnal tide
- Mixed tide
- Neap tide
- Semidiurnal tide
- Spring tide

Transition element

A special 'T' shaped 5 node element which makes the transition between a one dimensional element and a two dimensional element.

Turbulence

In a turbulent motion, the true velocity and pressure vary in a disorderly manner. A turbulent motion is always unsteady, since at a given point the velocity changes continuously in a very irregular way.

Turbulent exchange coefficient

A frequently applied approximation is derived from the assumption that the Reynolds stress varies linearly with the gradient of the time-averaged velocities. Under this theory, the stresses caused by random turbulent motions are analogous to Newton's law of viscosity for viscous stresses arising from molecular motions. This approximation gives rise to the turbulent coefficient of viscosity, also called eddy viscosity.

Two delta-X

A numerical instability which presents itself as a high value followed by a low value followed by a high value at the corners of the elements. When contoured, a two delta-X oscillation looks like a case of mesh measles.

Two dimensional element

A triangle (3 corners and 3 mid-side nodes) or quadrilateral (4 corners and 4 mid-side nodes) shape which defines the geometry in two space coordinates and averages over the third space coordinate. In a two dimensional *Horizontal* model, the averaging occurs over depth. In a two dimensional *Vertical* model, the averaging occurs over width. Several two dimensional horizontal elements aligned side by side may accurately define the bottom elevation of a navigation channel.

Verification

The process by which we gain confidence in the ability of our model to predict behavior of the prototype. Field data, like the model results, are only an approximation of reality and must be treated with skepticism. In verifying RMA2, the primary adjustments to be made are to the geometry, boundary conditions, roughness, and eddy viscosity. These adjustments are made interactively until the model agrees satisfactorily with field (prototype) observations.

Vertically homogeneous fluid

Refers to the similarity of the fluid within the water column.

Viscosity

The degree to which a fluid resists flow under an applied force.



See Also:

“Modeling Turbulence” on page 37.

Vorticity

Fluid flow involving rotation about an axis. A spiral motion of fluid within a limited area, especially a whirling mass of water or air that sucks everything near it toward its center.



See Also:

“Bendway Correction (Vorticity)” on page 74.

Water column

An elemental projection in the z direction. The water profile from the surface to the bottom of the water body.

Waterways Experiment Station

The US Army Engineer Waterways Experiment Station (WES), located in Vicksburg, Mississippi, is the principal research, testing, and development facility of the US Army Corps of Engineers. Its mission is to conceive, plan, study, and execute engineering investigations and research and development studies in support of civil and military missions of the Chief of Engineers and other federal agencies.

WES is composed of the following 6 laboratories:

Hydraulics Laboratory

Geotechnical Laboratory

Structures Laboratory

Environmental Laboratory
Coastal Engineering Research Center
Information Technology Laboratory

Weir

An obstruction placed in a stream, diverting the water through a prepared aperture for measuring the rate of flow.

Well formed element

An element with the proper aspect ratio, shape, angle, plane, and depth variation along an element (gradient).

Well-confined channel

A channel in which the water cannot escape the banks.

Word size

A Computer term. A word is made up of a group of bytes. A system's word size is defined by the number of bytes necessary to make a word on that particular type of computer system. For example, a typical PC uses a two byte word (16 bits), where the Cray YM-P uses an eight byte word (64 bits).

Index

- Absorption. *See* Boundary conditions, Reflection/Absorption
- AC1 variable, 52, 126
- AC1X variable, 110, 111
- AC2 variable, 52, 127
- AC2X variable, 111
- AC3 variable, 52, 127
- AC3X variable, 111
- AC4 variable, 52, 127
- AC4X variable, 111
- Adding elements. *See* Mesh, Modifying, Elements
- Adding nodes. *See* Mesh, Modifying, Nodes
- ALFAK variable, 104
- Alternate dynamic boundary conditions. *See* Files, Alternate dynamic boundary conditions
- AO variable, 142, 147
- Applications for RMA2, 1
- Area
 - of Elements. *See* Elements, Area of
- Array dimensions, 169
- ASC2BIN program, 195
- ASEC variable, 76, 168
- Averaging solutions. *See* Files, Solution, Averaging
- AVG CHG, 55
- Banners
 - T3 card, 160
- Batch mode, 16
- Bed friction, 36
 - Automatically assigning, 37, 77
 - HN card, 148
 - Manually assigning, 37
 - Problems when using, 47
- Bendway correction, 74
 - BV card, 113
 - Using, 76
 - VO card, 168
- BETA variable, 110
- BETA_L variable, 89, 120
- BETA_T variable, 89, 120
- BIN2ASC program, 195
- Boundary break angle, 17, 44, 46, 63, 72, 173
- Boundary conditions, 29
 - Advanced techniques, 77
 - Assignment tips, 35
 - BA card, 104
 - BCC card, 105
 - BCN card, 106
 - BH card, 107
 - BQ card, 108
 - BRA card, 110
 - BRC card, 111
 - BS card, 112
 - BV card, 113
 - BW card, 114
 - BWC card, 115
 - Current speed, 74
 - Direction of flow, 73
 - Dynamic, 30
 - Flow, 32
 - Total flow, 32
 - Velocity, 33
 - for vorticity, 76
 - In listing file, 52
 - Location of, 46
 - Oscillations, 32, 45
 - Parallel (slip) flow, 31
 - Forcing parallel flow, 32
 - Problems when using, 35, 46
 - Rating curve, 74
 - Reflection/Absorption, 74
 - REV card, 156
 - Revising, 73, 156
 - Stagnation point, 34
 - Types of, 31

Typical examples, 34
 Unsteady state. *See* Boundary conditions, Dynamic
 Water level (head), 33
 Buffer blocks, 54. *See* Files, Temporary

 Calculation variables, 90
 CA card, 122
 Capabilities of RMA2, 1
 Changes in RMA2 4.30, 8
 Check list (pre-run), 17
 Chezy. *See* Bed friction
 Comments, 42
 CO card, 123
 Computation time. *See* Simulation time
 Computer
 Specifying type, 23, 181
 \$M card, 103
 Conservation of mass. *See* Mass conservation
 Continuity
 GC card, 139
 GCL card, 140
 Continuity check lines
 Continuing, 185
 List of assigned, 51
 Continuity checks, 56
 Control structure elements. *See* Elements, Control structure
 FC card, 134
 Convergence. *See* Bendway correction
 Criterion for, 25, 26
 Meeting, 27
 with Vorticity, 168
 Parameters, 55
 Problems. *See* Divergence
 Unsatisfactory convergence, 27
 When hotstarting, 64
 CORD variable, 142
 Coriolis, 52, 85, 138
 LA card, 150
 Current speed. *See* Boundary conditions, Current speed
 Curving element edges. *See* Mesh, Modifying, Curving element edges

 Dam. *See* Elements, Control structure
 Data cards, 91, 100
 DECAY variable, 89
 DECAY variable, 120
 DELT variable, 105, 167
 Density of fluid, 77
 Dimensions, 50
 Direction of flow. *See* Boundary conditions, Direction of flow
 Discharge boundary
 BQ card, 108
 RA card, 153
 Discharging, 83. *See* Elements, Control structure
 Divergence, 17, 18, 25, 27, 55, 90
 when Wetting and drying, 79, 80
 Drying. *See* Wetting and drying
 DSEC variable, 76, 168
 DSET variable, 79, 124
 DSETD variable, 79, 124

 Eddy viscosity. *See* Turbulence, Automatically assigning. *See* Turbulence
 Obtaining guideline values for, 196
 Element connection table, 52, 70
 GE card, 141
 Element flow, 83
 Element types supported, 4
 Elemental elimination. *See* Wetting and drying, Elemental elimination
 Elements
 Area of, 52
 Control structure, 8, 62
 in Listing file, 53
 Curved edges, 8. *See* Mesh, Modifying, Curving element edges
 Junction, 7, 60
 Momentum, 7, 61
 Total head, 7, 61
 Water surface, 7, 61
 Material type, 7, 22
 Changing, 71
 GT card, 144
 with rainfall and evaporation, 84
 Modifying. *See* Mesh, Modifying, Elements
 One dimensional, 5, 59
 Properties, 16
 Removing
 via Material type, 72
 Transition, 6, 60
 Two dimensional, 5
 ELEV variable, 149
 Ending an RMA2 run, 40
 English units. *See* Units, Specifying
 EPSXX variable, 151
 EPSXY variable, 151
 EPSYX variable, 151
 EPSYY variable, 151
 Equations, 3
 Number of, 54
 Specifying for rating curve. *See* Boundary conditions, Rating curve
 Evaporation, 84
 RA card, 153
 Exit boundary, 33

FastTABS, 195

Files

- \$L card, 101
- Alternate dynamic boundary conditions, 19, 193
- Format of binary output, 191
- Geometry, 20
- Hotstart, 20
- Results listing, 20
 - Full, 21, 39
 - Customizing, 68
 - Described, 49
 - TR card, 163
 - When to write to, 68
 - Summary, 21
 - Described, 57
 - Obtaining, 69
 - Obtaining after run, 196
 - TRN card, 164
- Run control, 19
- Scratch. *See* Files, Temporary
- Solution, 21
 - Averaging, 196
 - Customizing, 67
 - Merging, 196
 - Moving across platforms, 195
 - Subtracting, 195
 - TO card, 162
 - TS card, 165
- Temporary, 169
 - Error concerning, 174
 - Number of, 54
 - Reducing number of, 170
- Vorticity, 22
 - TS card, 165

FLD variable, 113, 123, 129, 156, 158

Flow control. *See* Control structure elements

Flow direction. *See* Boundary conditions, Direction of flow

Fluid density, 77

- FD card, 136

Fluid properties, 77

Fluid temperature, 77

FLZ variable, 134

Friction. *See* Bed friction

Front width, 55

Frontal Passages. *See* Storms

Frontal storm passages

- BWS card, 120

Full print file. *See* Files, Results listing, Full

Full results listing. *See* Files, Results listing, Full

Gate. *See* Elements, Control structure

Geometry. *See* Files, Geometry

- G1 card, 138
- GC card, 139
- GCL card, 140
- GE card, 141
- GN card, 142
- GS card, 143
- GT card, 144
- GV card, 145
- GW card, 146
- GZ card, 147

Geometry File. *See* Files, Geometry

Geometry input summary, 51

GFGEN, 16, 17, 19, 20, 51, 67, 69, 95, 170, 174, 175, 185

Governing equations, 3

GPEC variable, 151

Head. *See* Boundary conditions, Water level (head)

Head boundary

- BH card, 107

Helical flow patterns. *See* Bendway correction. *See* Bendway correction

HMIN variable, 149

Hotstart File. *See* Files, Hotstart

Hotstarting, 64, 67

- \$L card, 101
- How to, 65
- Problems, 67, 175
- with Reflection/Absorption, 74

IBATCH variable, 14

IBHO variable, 165

IBUP variable, 101, 186, 193

IBVO variable, 77, 165

ICON variable, 110

IECHO variable, 163

IFINO variable, 101

IGEON variable, 101

IHOTN variable, 101

IHOTO variable, 101

IMAT variable, 141, 144, 150, 151

Inflow

- per Element. *See* Element flow

Inflow boundary

- RA card, 153

Initial conditions, 23, 25

- Effects of, 45
- Throwing out, 67

IC card, 149

in listing file, 53

When Coldstarting, 24

When Hotstarting, 24

Input variables, 94

Interactive mode, 14

IOUT variable, 101

IPEC variable, 151
 IPRT variable, 52, 53, 163
 IRUFF variable, 154
 ISPRT variable, 9, 69
 ISTART variable, 150, 154
 ISTYPE variable, 88, 120
 Iterations, 24
 Convergence criterion. *See* Convergence, Criterion
 for
 Explanation of, 24
 Specifying the number of, 25, 27
 TI card, 161
 Vorticity, 75, 76
 TV card, 166
 ITRACE variable, 163
 ITSI variable, 163, 176
 IVOR variable, 76, 168
 IVRSID variable, 103
 IWIND variable, 105, 115
 IWMX variable, 115
 IWR33 variable, 101

 JCR variable, 111
 Job sheet, 99
 Junction elements. *See* Elements, Junction

 King, Dr. Ian, 1

 Latitude. *See* Coriolis
 LHED, 188
 LI variable, 79, 124
 Limitations of RMA2, 2
 LINE variable, 139, 140
 LMT variable, 139

 Machine identifier, 51, 174, 181. *See* Computer,
 Specifying type
 Macintosh, 10
 Mainframe computer, 11
 MAKE_EV program, 38, 196
 Manning's equation. *See* Bed friction
 Marsh porosity. *See* Wetting and drying, Marsh
 porosity
 Marsh porosity parameters, 52
 Mass conservation, 61, 72
 Material type. *See* Elements, Material type
 for Junction elements, 61
 for Transition elements, 60
 MAX BUFFER SIZE, 50, 54
 MAX CHG, 55
 MAXIMUM FRONT WIDTH, 55
 MAXSTRM variable, 171
 MBAND variable, 65, 167
 MCC variable, 170, 184, 188

 MCCN variable, 170, 182, 188
 MEL variable, 170, 184, 185
 Memory. *See* System requirements
 MERGAVG program, 68, 196
 Merging solutions. *See* Files, Solution, Merging
 Mesh
 Design, 16
 Modifying, 69
 Bottom elevations, 72
 Curving element edges, 72
 Element eddy viscosity tensor, 72
 Elements, 70
 Adding, 70
 Removing, 71
 Nodes, 70
 Adding, 70
 Moving, 70
 Resizing the mesh, 72
 Properties, 17
 Resolution, 8, 17, 44, 46
 Continuity, 44
 Parallel flow angle, 32
 Metric. *See* System International
 Metric units. *See* Units, Specifying
 METRIC variable, 157
 MFW variable, 170
 Mini computer, 11
 MND variable, 170, 184, 185
 Model information, 50
 Modification date, 50
 MPB variable, 171
 MR1 variable, 170
 MXSTRM variable, 181

 Navigation. *See* Ship simulator
 NB variable, 65
 NBC array, 188
 NBS variable, 170, 171
 NBX variable, 9, 105, 138
 NCYC variable, 167
 NDRY variable, 58
 Newton-Raphson convergence scheme, 24
 NFIX variable, 106
 NGOODMAX variable, 76, 166
 NITI variable, 65, 161
 NITN variable, 79, 161
 NJN variable, 134
 NJT variable, 134
 Nodal results, 57
 Nodal specifications
 in listing file, 53
 Nodes
 Modifying. *See* Mesh, Modifying, Nodes
 NODES ACTIVE, 56

NOP variable, 141
 NOPTV variable, 77, 101
 NPASS1 variable, 76, 166
 NPASS2 variable, 76, 166
 NREF variable, 89, 120
 NSPLPT variable, 164
 NSTART variable, 65, 167, 175
 Numerical oscillations. *See* Boundary conditions, Oscillations
 NVITI variable, 76, 166
 NVITN variable, 76, 166

 OMEGA variable, 138, 150
 One dimensional elements. *See* Elements, One dimensional
 Origin of the program, 1
 ORT variable, 130, 148
 Oscillating
 Boundary conditions, 45
 Oscillations, 44
 Two delta x, 47
 when Wetting and drying, 79, 80
 Outflow
 per Element. *See* Element flow

 PC computer, 10
 Peclet. *See* Turbulence, Peclet. *See* Turbulence, Automatically assigning
 Peclet formula, 39. *See* Turbulence, Peclet formula
 Performance, 169
 Permeability of boundary. *See* Boundary conditions, Reflection/Absorption
 Ponds. *See* Wetting and drying, Elemental elimination, Ponds
 Problems using RMA2, 173
 Proclv, 16
 Pumping, 83. *See* Elements, Control structure

 QDIR variable, 109
 QF variable, 108
 QVEC variable, 108
 QXP variable, 9, 109

 R2_2_SUM program, 196
 R2DIFF program, 195
 Rainfall, 84
 RA card, 153
 Rating curve. *See* Boundary conditions, Rating curve
 BRC card, 111
 Example, 111
 RCMIN variable, 76, 168
 RDCOEF variable, 154
 RDD0 variable, 154
 RDR0 variable, 154
 RDRM variable, 154
 Redimensioning RMA2, 170
 Reflection. *See* Boundary conditions, Reflection/Absorption
 Resolution. *See* Mesh, Resolution
 Restarting. *See* Files, Hotstart
 Results
 Interpretation of, 49
 Results listing control, 39
 Results listing files. *See* Files, Results listing
 Resuming a simulation. *See* Hotstarting
 Revising. *See* Boundary conditions, Revising
 Revisions to time step
 In listing file, 57
 RMA4, 22, 44, 77, 101
 RON variable, 136
 Roughness. *See* Bed friction
 RD card, 154
 Run control, 91
 Run control file. *See* Files, Run control
 Run control parameters, 51

 Scale factor
 with Eddy viscosity. *See* Turbulence, Scale factors
 Scale factors. *See* Geometry, Scale factors
 GS card, 143
 Scratch files. *See* Files, Temporary
 SED2D, 44
 Ship simulator studies, 17
 SI units. *See* Units, Specifying
 SIDF variable, 84, 108, 153
 Simulation progress and statistics, 54
 Simulation time, 29
 Dynamic, 29
 Steady state, 29
 Slip flow. *See* Boundary conditions, Parallel (slip) flow
 Solution, 4
 Guidelines for a good one, 16
 Interpretation of, 58
 Solution File. *See* Files, Solution
 Solution matrix. *See* Files, Temporary
 Source code, 10, 14, 169, 170
 SPEC variable, 106, 107
 Spin-up, 64, 67. *See* Verifying the simulation, Effects of initial conditions
 SS1 variable, 142, 146
 SS2 variable, 142, 146
 SSDCRT variable, 79, 161
 SSPD variable, 89, 120
 SSVCRT variable, 76, 168
 SSWSE variable, 110
 Starting conditions. *See* Initial conditions
 Stopping the simulation, 42
 STO card, 158

- Storms, 86
 - BWS card, 120
 - Defining Storms, 88
- STREMR numerical model, 75
- Subtracting solutions. *See* Files, Solution, Subtracting
- Summary print file. *See* Files, Results listing,
 - Summary
- Summary results listing. *See* Files, Results listing,
 - Summary
- Super computer, 11
- Support. *See* Technical support
- System International, 157
- System requirements, 10

- TAREA variable, 110
- TAX variable, 114
- Taylor series, 90
- TBINRY variable, 162
- Technical support, 197
- TEMP variable, 137
- Temperature. *See* Fluid temperature
 - FT card, 137
- Temporal derivative calculation, 90
- Temporary files. *See* Files, Temporary
 - Number of, 54
- Terminating RMA2. *See* Stopping the simulation
- TH variable, 141, 145
- Theory
 - Bendway correction (vorticity), 75
 - Governing equations, 3
 - Storms, 86
- THETA variable, 110, 111
- THETA_S variable, 90, 120
- THETA_T variable, 90, 120
- Tidal cycle, 29
- Tides, Simulating with, 29
- Time. *See* Simulation time
- Time step
 - Controlling, 28
 - TZ card, 167
 - END card, 129
 - In the solution file, TS card, 165
 - Selecting interval, 28
 - Steady state simulation, 28
- TITLE variable, 159, 160
- Titles, 22
 - End of, 22
 - T1-T2 cards, 159
 - T3 card, 160
- TMAX variable, 167
- Transition elements. *See* Elements, Transition
- TREF variable, 89, 120
- Turbulence, 37, 52
 - Automatically assigning, 39, 78
 - EV card, 130
 - EX card, 132
 - EY card, 133
 - GV card, 145
 - Manually assigning, 38
 - PE card, 151
 - Peclet, 151
 - Peclet formula, 39
 - Problems when using, 47
 - Ranges of turbulent exchange coefficients, 38
 - Scale factors, 151
 - Specifying, 38
- Two dimensional elements. *See* Elements, Two dimensional
- TWX variable, 114

- Units
 - SI card, 157
 - Specifying, 67
- UNOM variable, 149
- USDCRIT variable, 79
- USDCRT variable, 161
- USDVCC variable, 76, 168
- USERCA1 variable, 122

- Variables
 - Input, 94
- Verifying the simulation, 43
 - Continuity, 44
 - Continuity check lines, 44
 - Critical check points, 44
 - Description of the process, 43
 - Effects of initial conditions, 45
 - Influences on, 47
 - Problems, 45
 - Wetting and drying concerns, 45
- Version number, 50
- Vertical circulation, 87
- VOR variable, 113
- Vorticity. *See* Bendway correction
- VPEC variable, 151
- VVEC1 variable, 112
- VVEC2 variable, 112

- Water surface elevation. *See* Boundary condition,
 - Water level (head)
- WC1 variable, 115
- WC2 variable, 115
- WC3 variable, 115
- WC4 variable, 115
- Wetting and drying, 78
 - Boundary condition problems, 36
 - DE card, 124
 - DM card, 126

- Elemental elimination, 79
 - Advantages of, 79
 - Disadvantages of, 80
 - Frequency of checking, 79
 - Ponds, 79
 - Problems, 80
- In listing file, 56
- Marsh porosity, 80
 - Advantages of, 82
 - Disadvantages of, 82
 - Frequency of checking, 82
 - Problems, 82
- WIDS variable, 142, 146
- WIDTH variable, 142, 146
- Wind, 85. *See* Storms
 - BW card, 114
 - BWC card, 115
 - Ekman Formula, 118
 - Generic Formula, 118
 - RMA2 Original Formula, 118
 - Safaie Formula, 118
 - Van Dorn Formula, 118
 - Wu Formula, 118
- WNDMAX variable, 89, 120
- WNDMIN variable, 89, 120
- Workstation, 11
- XSCALE variable, 138, 143
- YSCALE variable, 138, 143
- ZSCALE variable, 143

Notes

